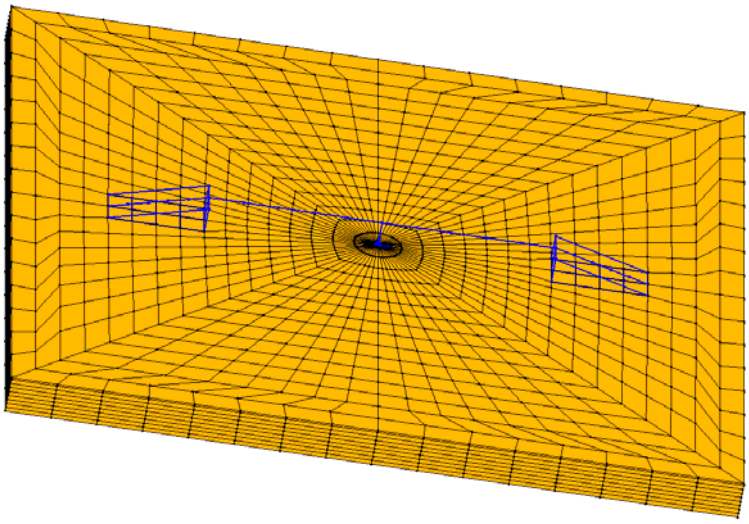
BridgePBEE: OpenSees 3D Pushover and Earthquake Analysis of Single-Column 2-span Bridges

User Manual (Beta 1.0)



http://peer.berkeley.edu/bridgepbee/

Jinchi Lu, Kevin Mackie, and Ahmed Elgamal

December 2011

Table of Contents

1 Introduction 1
1.1 Overview 1
1.2 System Requirements 2
1.3 Acknowledgments 3
1.4 Units 4
2 Getting Started 5
2.1 Start-Up 5
2.2 Interface 5
2.2.1 Menu Bar 6
2.2.2 Model Input Window 7
2.2.3 Finite Element Mesh Window 8
3 Bridge Model 9
3.1 Column Parameters 10
3.1.1 Column Linear Material Properties 11
3.1.2 Nonlinear Fiber Section 12
3.1.3 Column below Grade 17
3.2 Bridge Deck Parameters 17
3.3 Embankment Parameters 18
3.4 Abutment Parameters 19
3.4.1 Elastic Abutment 19
3.4.2 Roller Model 21
3.4.3 Simplified Model (SDC 2004) 22
3.4.4 Spring Model 26
3.4.5 SDC (2010) Sand 30
3.4.6 SDC (2010) Clay 30
3.4.7 EPP-Gap Model 30
3.4.8 HFD Model 32
4 Soil Parameters 32
5 Pushover & Eigenvalue Analyses 37
5.1 Load Pattern 37
5.1.1 Pushover by User-Defined Load Pattern (U-Push) 38
5.2 Output for Pushover Analysis 40
5.2.1 Column Response Time Histories and Profiles 41
5.2.2 Column Response Relationships 41
5.2.3 Abutment Force-Displacement and Response Time Histories 42
5.2.4 Deformed Mesh 42
5.3 Eigenvalue Analysis 43
6 PBEE Analysis (Ground Shaking) 45
6.1 Theory and Implementation of PBEE Analysis 45
6.2 Input Necessary for User-defined PBEE Quantities 48
6.3 Definition/specification of PBEE input motion ensemble (suite) 49
6.3.1 Available Ground Motions 49
6.3.2 Specifications of PBEE Input Motions 50

6.4 Save Model and Run Analysis 54
6.5 PBEE Analysis 55
6.5.1 PBEE Quantities 56
6.5.2 Compute Repair Cost & Time 58
6.5.3 Compute Hazard Curves 59
6.5.4 Compute Disaggregation 60
7 Time History and PBEE Output 61
7.1 Time History Output Quantities 61
7.1.1 Column Response Time Histories and Profiles 64
7.1.2 Column Response Relationships 70
7.1.3 Abutment Responses Time Histories 73
7.1.4 Deformed Mesh 76
7.1.5 Soil Response Time Histories 77
7.1.6 PBEE Output Quantities 81
7.1.7 Bridge Peak Accelerations for All Motions 88
7.1.8 Maximum Column & Abutment Forces for All Motions 90
7.2 PBEE Outcomes 93
7.2.1 Repair Cost & Time 93
7.2.2 Hazard Curves 96
7.2.3 Disaggregation 100
7.2.4 EPS Version of All PBEE Figures 102
8 Appendix A: How to Define the Soil Finite Element Mesh 104
9 Appendix B: Simple Pushover Examples (Bridge on Rigid Ground) 113
10 Appendix C: How to Incorporate User-defined Motions 118
11 Appendix D: Calculation of Steel and Concrete Material Properties 122
12 Appendix E: Customization of PBEE Quantities 127
13 References 132

List of Figures

Fig. 1. Coordinate system in BridgePBEE 3
Fig. 2. BridgePBEE main window 5
Fig. 3. BridgePBEE's menu and submenu bars: a) menu bar; b) menu File; c) menu Execute; d) menu Display; and e) menu Help 6
Fig. 4. BridgePBEE copyright and disclaimer window 7
Fig. 5. Buttons available in the Finite Element Mesh window 8
Fig. 6. BridgePBEE main window (defining a bridge model) 9
Fig. 7. Bridge Model window 10
Fig. 8. Steps to define the elastic properties of the column 11
Fig. 9. Steps to define a nonlinear Fiber Section 12
Fig. 10. Fiber Section window 13
Fig. 11. Column fiber section (based on PEER best modeling practices report, Berry and Eberhard, 2007) 13
Fig. 12. Stress-strain curve for Steel02 material (default values employed; "C" represents compression and "T" represents tension) 15
Fig. 13. Stress-strain curve of Concrete02 material for the core concrete (default values employed; "C" represents compression and "T" represents tension) 15
Fig. 14. Stress-strain curve of Concrete02 material for the cover concrete (default values employed; "C" represents compression and "T" represents tension) 16
Fig. 15. Moment-curvature response for the column (with default steel and concrete parameters, and the deck weight 11,915 kN applied at the column top) 16
Fig. 16. Steps to define the deck geometrical configuration and material properties 17
Fig. 17. Elastic abutment model 19
Fig. 18. Steps to define Elastic abutment model 20
Fig. 19. Longitudinal force-displacement relationship for the Elastic abutment model 20
Fig. 20. Roller abutment model 21
Fig. 21. Steps to define a Roller abutment model 21
Fig. 22. Longitudinal force-displacement relationship for the Roller abutment model 22
Fig. 23. General scheme of the Simplified abutment model (Aviram *et al*., 2008) 22
Fig. 24. Longitudinal backbone curve force-displacement relationship (two on each end of the bridge; Caltrans SDC, 2004) 23
Fig. 25. Steps to define the Simplified abutment model. 24
Fig. 26. Longitudinal force-displacement relationship for the Simplified abutment model: a) longitudinal direction; b) transverse direction 25
Fig. 27. General scheme of the Spring abutment model (Aviram et al. 2008) 27
Fig. 28. Steps to define a Spring abutment model 28
Fig. 29. Force-displacement relationship for the Spring abutment model: a) longitudinal direction; b) transverse direction. 29
Fig. 30. Steps to define a SDC 2010 Sand abutment model 30
Fig. 31. Steps to define a SDC 2010 Clay abutment model 31
Fig. 32. Steps to define an EPP-Gap abutment model 31
Fig. 33. Steps to define a HFD abutment model: a) choosing HFD model at the bridge model window; b) HFD model window 33

Fig. 34. Analysis options 34
Fig. 35. Beam-column element types available for column 34
Fig. 36. Rayleigh damping coefficients 35
Fig. 37. Soil strata definition 35
Fig. 38. User-defined clay material U-Clay2 36
Fig. 39. Steps to define a load pattern for pushover analysis 37
Fig. 40. Load pattern for pushover analysis 38
Fig. 41. Steps to define a user-defined load pattern (U-Push) 39
Fig. 42. Example of user-defined pushover load pattern (U-Push) 40
Fig. 43. Response time histories and profiles for column (and pile shaft) 41
Fig. 44. Response relationships for column (and pile shaft) 41
Fig. 45. Abutment response time histories 42
Fig. 46. Deformed mesh and contour fill 42
Fig. 47. Steps to perform an Eigenvalue analysis 43
Fig. 48. Sample output for an Eigenvalue analysis 44
Fig. 49. Schematic procedure of the LLRCAT methodology for a single bridge component 46
Fig. 50. Steps to define PBEE motions 51
Fig. 51. PBEE input motions widow 52
Fig. 52. Intensity measures, time histories and response spectra of individual record 53
Fig. 53. Histogram and cumulative distribution for the whole input motion set 53
Fig. 54. Intensity Measures (IM) table for the whole input motion set 54
Fig. 55. Options to change number of records to be run at the same time 54
Fig. 56. OpenSees analysis in progress 55
Fig. 57. PBEE analysis window 55
Fig. 58. Damage states window 56
Fig. 59. Repair quantities window 57
Fig. 60. Unit Costs window 57
Fig. 61. Production Rates window 58
Fig. 62. Post-processing capabilities (menu options) available in a pushover analysis 62
Fig. 63. Post-processing capabilities (menu options) available in a base shaking analysis 62
Fig. 64. Analysis options in BridgePBEE 63
Fig. 65. Steps to display output for a different input motion: a) click menu Display (Fig. 3); b) select an input motion 64
Fig. 66. Menu items to access the column response time histories and response profiles 66
Fig. 67. Response time histories and profiles for column (and pile shaft): displacement is shown at the nodes (only one element is used above ground). 67
Fig. 68. Bending moment profile in the longitudinal plane 67
Fig. 69. Response summary 68
Fig. 70. Column longitudinal displacement response time histories: a) response profiles at specific load steps; b) response time histories at different elevations 69
Fig. 71. Column longitudinal acceleration response time histories at different elevations (free-field and input accelerations are also included) 70
Fig. 72. Menu items to access the column response relationships 71
Fig. 73. Column response output options 71
Fig. 74. Load-displacement curve at column top 72
Fig. 75. Moment-curvature curve at column top 72

Fig. 76. Menu items to access the abutment responses 73
Fig. 77. Menu items to access the abutment responses 74
Fig. 78. Abutment response time histories (scroll down to see all directions): a) abutment force-displacement relationships; b) relative deck-end/abutment displacement time histories; c) resisting force time histories; and d) abutment pile cap time histories 75
Fig. 79. Menu items to access the deformed mesh 76
Fig. 80. Deformed mesh 77
Fig. 81. Menu items to access the soil responses 78
Fig. 82. Response options for soil time histories 79
Fig. 83. Planes for locations of the soil response time histories 79
Fig. 84. Locations of soil response time histories 80
Fig. 85. Soil settlement time histories under abutment 80
Fig. 86. Finite element mesh in BridgePBEE: Node O – Column base node (at ground surface); Node A – Column top node; Node B – Deck-end node; Node C – Abutment top node (having the same coordinates as Node B); Nodes B are C are connected by an abutment model; Node D – Abutment pile cap node 82
Fig. 87. Code snippet to calculate the tangential drift ratio of column 83
Fig. 88. Menu items to access the PG quantities for all motions 85
Fig. 89. PG quantities for all motions (scroll down to see all 11 PGs) 86
Fig. 90. Converting figures to EPS format 86
Fig. 91. Lognormal standard deviations (beta values) for each PG: a) table format; b) bar graph format 87
Fig. 92. Menu items to access bridge peak accelerations for all motions 88
Fig. 93. Bridge peak accelerations for all motions: a) maximum bridge accelerations; b) maximum column base accelerations; and c) maximum free-field accelerations 90
Fig. 94. Menu items to access maximum column & abutment forces for all motions 91
Fig. 95. Maximum column & abutment forces for all motions: a) maximum column shear forces; b) maximum column bending moments; and c) maximum abutment forces 92
Fig. 96. Contribution to expected repair cost ($) from each performance group 93
Fig. 97. Total repair cost ratio (%) as a function of intensity 94
Fig. 98. Contribution to expected repair cost ($) from each repair quantity 94
Fig. 99. Contribution to repair cost standard deviation ($) from each repair quantity 95
Fig. 100. Total repair time (CWD: Crew Working Day) as a function of intensity 95
Fig. 101. Contribution to expected repair time (CWD) from each repair quantity 96
Fig. 102. Mean annual frequency of exceedance (ground motion) 97
Fig. 103. Return period against total repair cost ratio 98
Fig. 104. Mean annual frequency of exceedance (loss) against total repair cost ratio 98
Fig. 105. Return period against total repair time 99
Fig. 106. Mean annual frequency of exceedance (loss) against total repair time 99
Fig. 107. Disaggregation of expected cost by performance group 100
Fig. 108. Disaggregation of expected repair cost by repair quantities 101
Fig. 109. Disaggregation of expected repair time by repair quantities 102
Fig. 110. Converting all PBEE figures to EPS format 103
Fig. 111. Schematic view of an idealized single bent bridge system 104
Fig. 112. General meshing controlling parameters (default values) 105
Fig. 113. Meshing controlling parameters for horizontal direction (default values) 106

Fig. 114. Adjusting mesh near embankment: a) before adjusting; b) after adjusting 107
Fig. 115. Meshing controlling parameters for vertical direction (default values) 108
Fig. 116. Finite element mesh created with default values 109
Fig. 117. Mesh refinement example 1: a) Change "Num of Slices" to 32; b) the resulting mesh 110
Fig. 118. Mesh refinement example 2: a) Change "Number of Mesh Layers" in the vertical direction; b) the resulting mesh 111
Fig. 119. Mesh refinement example 3: a) Change meshing controlling parameters in the horizontal direction; b) the resulting mesh 112
Fig. 120. Cantilever beam simulation using BridgePBEE 114
Fig. 121. Fixed-end beam simulation using BridgePBEE 115
Fig. 122. Deck deformation under gravity (the maximum displacement is 0.0372 m) 116
Fig. 123. Fixed end-roller beam analytical solution (from efunda.com) 116
Fig. 124. Longitudinal pushover 117
Fig. 125. Transverse pushover 117
Fig. 126. Choosing PBEE motion set 119
Fig. 127. Directory structure of PBEE motion set 120
Fig. 128. Sample .info file 120
Fig. 129. Sample .data file 121
Fig. 130. Example of user-defined motion 121
Fig. 131. Visual Studio file PBEE.SLN 128
Fig. 132. Modifying file PBEE.CPP 129
Fig. 133. Building PBEE.DLL in Visual Studio 2005 130
Fig. 134. Replacing file PBEE.DLL under the installation folder 131

List of Tables

Table 1. Default Values for Column RC Section Properties 14
Table 2. Default Values for Steel02 Material Properties 14
Table 3. Default values for Concrete02 Material Properties 14
Table 4. Default Values for Bridge Deck 18
Table 5. Default Values for Deck Material Properties 18
Table 6. Geometric and Material Properties of a Bearing Pad 27
Table 7. Spring Abutment Model Properties 28
Table 8. Abutment Configurations 28
Table 9. Clay material properties 36
Table 10. PBEE Repair Quantities 59
Table 11. PBEE Performance Groups 81

1 Introduction

1.1 Overview

BridgePBEE is a PC-based graphical pre- and post-processor (user-interface) for conducting Performance-Based Earthquake Engineering (PBEE) studies for bridge-ground systems. The user interface allows for:

- Management of ground motions
- Simplified structure and soil mesh generation
- Simplified assignment of material properties for both the soil and structure
- Time history and PBEE analyses
- Visualization of output data

The interface is unique because it enables complete PBEE studies in a single GUI-driven package. The PBEE implementation employed is based on Pacific Earthquake Engineering Research (PEER) Center's performance-based earthquake engineering framework (Cornell and Krawinkler, 2000). The framework includes several building blocks (intermediate probabilistic models) that allow the user to generate probabilistic estimates of repair cost and repair time (consequences or decision variables) directly. These results are obtained seamlessly in the interface alongside more traditional outputs such as displacements, strains, etc.

The intermediate models require:

- Hazard model that uses earthquake ground motion data to determine an intensity measure (IM)
- Demand model uses response from dynamic analysis to determine an engineering demand parameter (EDP)
- Damage model connects the EDP to a damage measure (DM) or discrete set of damage states (DS)
- Repair model describes repair methods and repair quantities (Q) necessary to return the DSs to original functionality
- Loss model links Qs to consequences that are termed the decision variables (DV). Repair cost and repair time can be thought of as two possible decision variable (DV) outcomes characterized probabilistically by the framework.

The models are required for each performance group (PG). PGs represent a collection of structural components that act as a global-level indicator of structural performance and that contribute significantly to repair-level decisions. Performance groups are not necessarily the same as load-resisting structural components. The complete analysis is accomplished using the local linearization repair cost and time methodology (LLRCAT), detailed more in Chapter 6. The interface handles all of the above-mentioned intermediate models and provides default data for the case of reinforced concrete box girder bridges.

The decision variables that can be generated as output are the repair cost ratio (RCR), or the ratio of repair cost to replacement cost, and the repair time (RT) or repair effort, measured in terms of

crew working days (CWD). These outcomes are presented graphically as loss models conditioned on earthquake intensity. In addition, site-specific ground motion hazard can be specified, and the user-interface will then also generate loss hazard curves (mean annual frequencies of exceeding different loss levels). The loss hazard curves are presented graphically as mean annual frequencies or return periods.

An important feature of the interface is that the PBEE analysis can be executed sequentially: ground motion selection, time history analysis, loss modeling, hazard, and visualization. However, once a final selection of geometry and materials has been made (the FEA model is not changing), the time history analyses do not need to be repeated. These are the most time and resources intensive portions of the complete analyses. Once the time history results are computed, the user may perform what-if scenarios by changing any of the parameters of the intermediate damage, loss, and hazard models. The PBEE portions of the analysis do not require recomputing the time history results unless the model is changed or a new selection of ground motions is made.

Finite element computations are conducted using OpenSees (http://opensees.berkeley.edu, Mazzoni et al. 2009), an open source framework developed by the Pacific Earthquake Engineering Research (PEER) Center. The current version of the interface is limited to ordinary bridge overpasses with two spans and a single-column bent. The analysis options available in **BridgePBEE** include:

- Pushover Analysis
- Modal Analysis
- Single 3D Base Input Acceleration Analysis
- Full Performance-Based Earthquake Engineering (PBEE) Analysis

This document describes how to conduct the above analyses in BridgePBEE. For information on how to download and install BridgePBEE, please visit the BridgePBEE website (http://peer.berkeley.edu/bridgepbee/).

The coordinate system employed in BridgePBEE is shown in Fig. 1. The origin is located at the column base (the ground surface).

1.2 System Requirements

BridgePBEE runs on PC compatible systems using Windows (NT V4.0, 2000, XP, Vista or Windows 7). The system should have a minimum hardware configuration appropriate to the particular operating system.

Internet Explorer 3.0 or above (or compatible Browser) with Java Applet enabled is needed to view the graphic results. For best results, your system's video should be set to 1024 by 768 or higher.

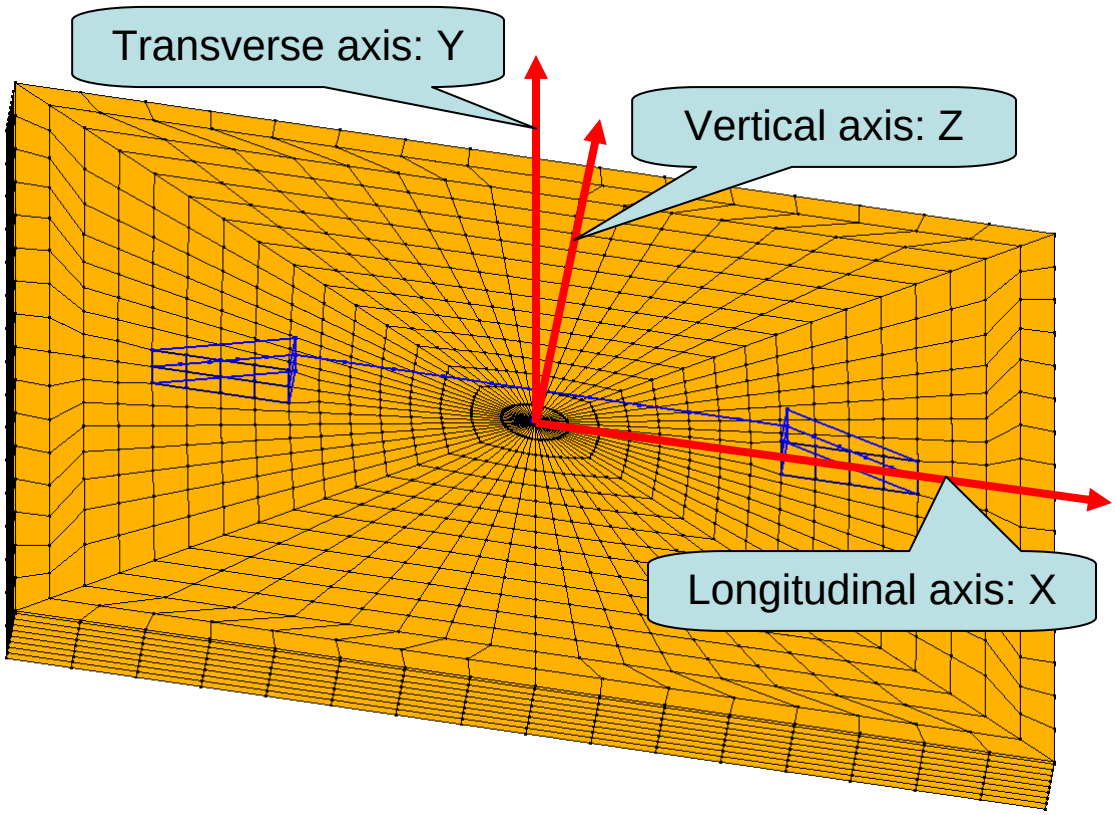


Fig. 1. Coordinate system in BridgePBEE

1.3 Acknowledgments

This research was funded in part by the Pacific Earthquake Engineering Research (PEER) Center Transportation Program (Dr. Stephen Mahin, Director and Dr. Yousef Bozorgnia, Executive Director). Dr. Steven Kramer (U. of Washington, Seattle) coordinated a review process of BridgePBEE. Feedback and suggestions provided by Dr. Kramer and the anonymous reviewers are greatly appreciated.

OpenSees (currently ver. 2.1.0 is employed) is a software framework (Mazzoni et al. 2009) for developing applications to simulate the performance of structural and geotechnical systems subjected to earthquakes (for more information, visit http://opensees.berkeley.edu/). Throughout, Dr. Frank McKenna was always generous and gracious with his assistance in matters related to OpenSees. The employed OpenSees geotechnical simulation capabilities were developed by Dr. Zhaohui Yang and Dr. Ahmed Elgamal. For more information, please visit http://cyclic.ucsd.edu/opensees/. The implemented PBEE analytical framework is provided by Dr. Kevin Mackie (U. Central Florida). For questions or remarks, please send email to Dr. Jinchi Lu (jinlu@ucsd.edu), Dr. Kevin Mackie (Kevin.Mackie@ucf.edu), or Dr. Ahmed Elgamal (elgamal@ucsd.edu).

BridgePBEE is written in Microsoft Visual C++ 2005 with Microsoft Foundation Class (MFC) Library version 8.0. The Java Applet package used to display graphical results in **BridgePBEE**

is obtained from the website http://ptolemy.eecs.berkeley.edu/. GIF images are generated with GNUPLOT for MS-Windows 32 bit Version 3.7, available at http://www.gnuplot.org/.

1.4 Units

The SI unit system is used throughout the user interface. For conversion between SI and English Units, please check:
http://www.unit-conversion.info/

Some commonly used quantities can be converted as follows:

- 1 kPa = 0.14503789 psi
- 1 psi = 6.89475 kPa
- 1 m = 39.37 in
- 1 in = 0.0254 m

2 Getting Started

2.1 Start-Up

On Windows, start **BridgePBEE** from the Start button or from an icon on your desktop. To Start **BridgePBEE** from the Start button:

1. Click Start, and then select Programs.
2. Select the **BridgePBEE** folder
3. Click on **BridgePBEE**

The **BridgePBEE** main window is shown in Fig. 2.

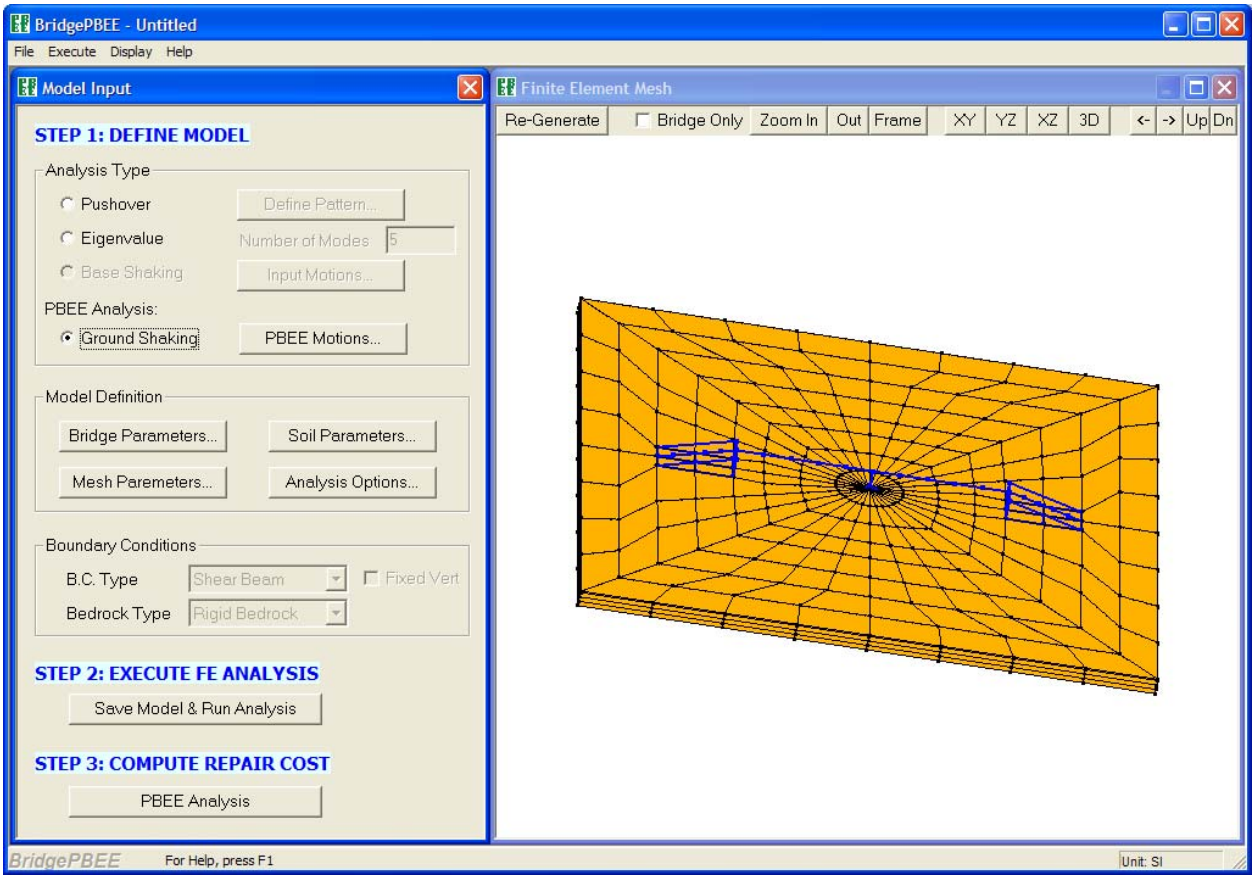


Fig. 2. BridgePBEE main window

2.2 Interface

There are 3 main regions in the **BridgePBEE** window – menu bar, the model input window, and the finite element mesh window.

2.2.1 Menu Bar

The menu bar, shown in Fig. 3, offers rapid access to most **BridgePBEE** main features.

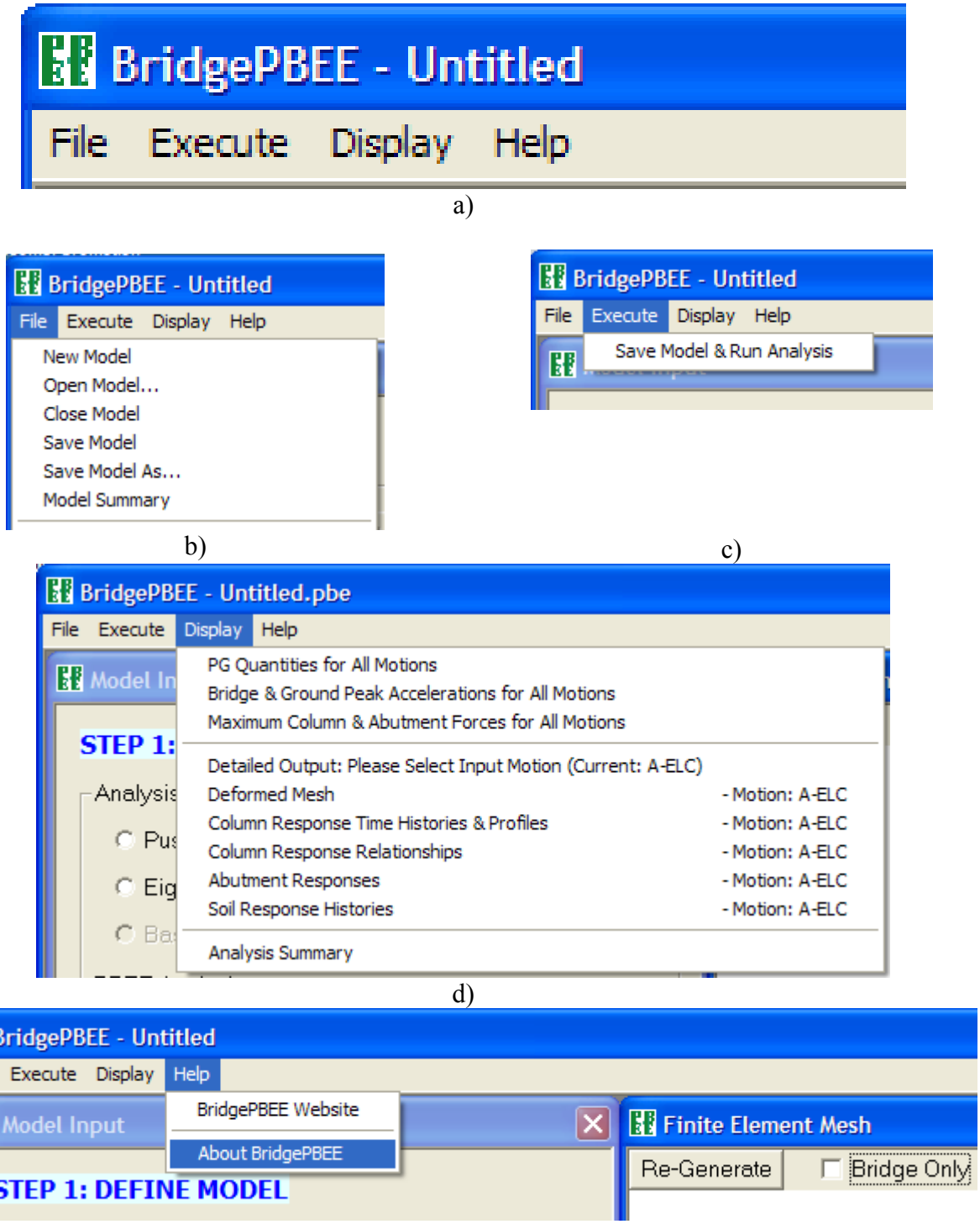


Fig. 3. BridgePBEE's menu and submenu bars: a) menu bar; b) menu File; c) menu Execute; d) menu Display; and e) menu Help

BridgePBEE's main features are organized into the following menus:

- **File:** Controls reading, writing and printing of model definition parameters, and exiting **BridgePBEE**.
- **Execute:** Controls running analyses.
- **Display:** Controls displaying of the analysis results.
- **Help:** Visit the **BridgePBEE** website and display the copyright/Disclaimer message (Fig. 4).

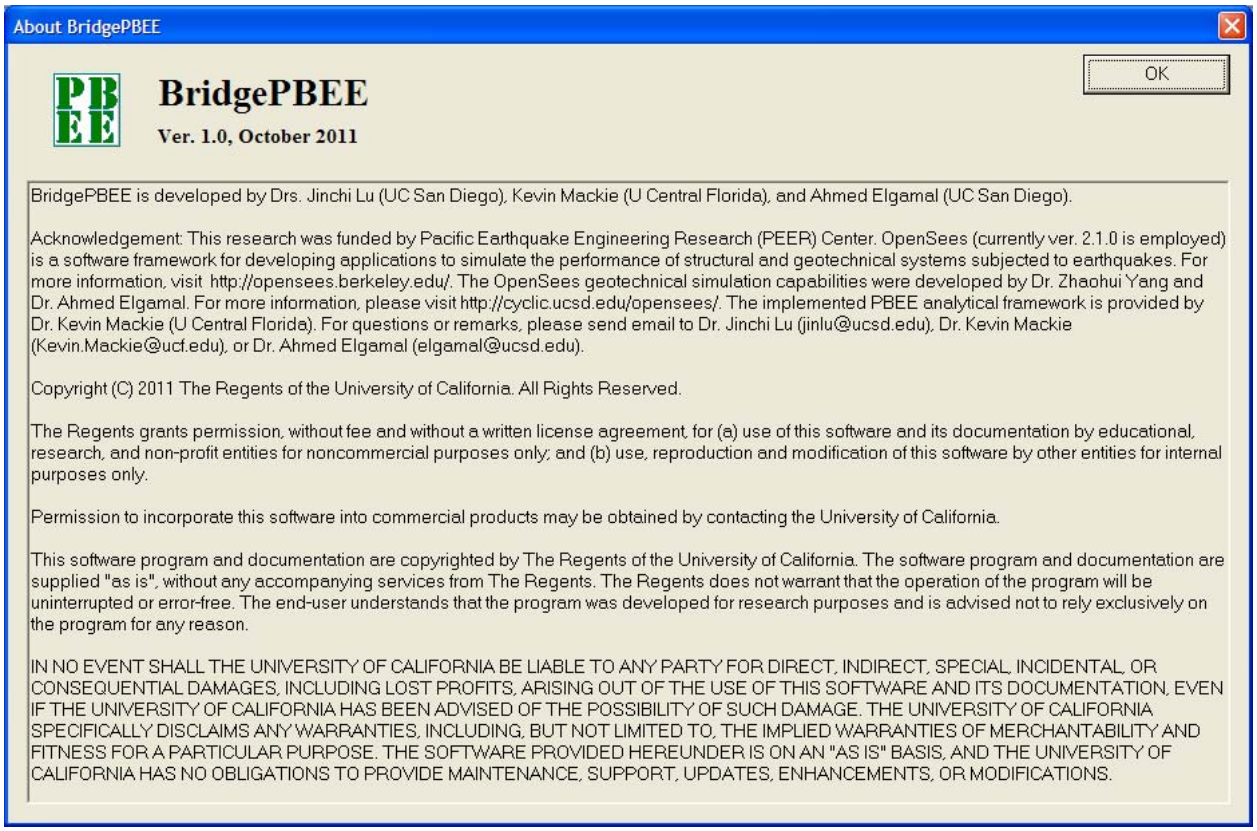

About BridgePBEE
OK
BridgePBEE
Ver. 1.0, October 2011
BridgePBEE is developed by Drs. Jinchi Lu (UC San Diego), Kevin Mackie (U Central Florida), and Ahmed Elgamal (UC San Diego).
Acknowledgement: This research was funded by Pacific Earthquake Engineering Research (PEER) Center. OpenSees (currently ver. 2.1.0 is employed) is a software framework for developing applications to simulate the performance of structural and geotechnical systems subjected to earthquakes. For more information, visit http://opensees.berkeley.edu/. The OpenSees geotechnical simulation capabilities were developed by Dr. Zhaohui Yang and Dr. Ahmed Elgamal. For more information, please visit http://cyclic.ucsd.edu/opensees/. The implemented PBEE analytical framework is provided by Dr. Kevin Mackie (U Central Florida). For questions or remarks, please send email to Dr. Jinchi Lu (jinlu@ucsd.edu), Dr. Kevin Mackie (Kevin.Mackie@uct.edu), or Dr. Ahmed Elgamal (elgamal@ucsd.edu).
Copyright (C) 2011 The Regents of the University of California. All Rights Reserved.
The Regents grants permission, without fee and without a written license agreement, for (a) use of this software and its documentation by educational, research, and non-profit entities for noncommercial purposes only; and (b) use, reproduction and modification of this software by other entities for internal purposes only.
Permission to incorporate this software into commercial products may be obtained by contacting the University of California.
This software program and documentation are copyrighted by The Regents of the University of California. The software program and documentation are supplied "as is", without any accompanying services from The Regents. The Regents does not warrant that the operation of the program will be uninterrupted or error-free. The end-user understands that the program was developed for research purposes and is advised not to rely exclusively on the program for any reason.
IN NO EVENT SHALL THE UNIVERSITY OF CALIFORNIA BE LIABLE TO ANY PARTY FOR DIRECT, INDIRECT, SPECIAL, INCIDENTAL, OR CONSEQUENTIAL DAMAGES, INCLUDING LOST PROFITS, ARISING OUT OF THE USE OF THIS SOFTWARE AND ITS DOCUMENTATION, EVEN IF THE UNIVERSITY OF CALIFORNIA HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGE. THE UNIVERSITY OF CALIFORNIA SPECIFICALLY DISCLAIMS ANY WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE. THE SOFTWARE PROVIDED HEREUNDER IS ON AN "AS IS" BASIS, AND THE UNIVERSITY OF CALIFORNIA HAS NO OBLIGATIONS TO PROVIDE MAINTENANCE, SUPPORT, UPDATES, ENHANCEMENTS, OR MODIFICATIONS.

Fig. 4. BridgePBEE copyright and disclaimer window

2.2.2 Model Input Window

The model input window controls definitions of the model and analysis options, which are organized into three regions (Fig. 2):

- **Step 1: Define Model:** Controls analysis types (pushover analysis, eigenvalue analysis or ground shaking) and analysis options; also controls definitions of bridge and soil strata including material properties. Meshing parameters are also defined.

- **Step 2: Execute FE Analysis:** Controls execution of the finite element analysis and display the progress bar.
- **Step 3: Compute Repair Cost:** Controls the PBEE analysis.

2.2.3 Finite Element Mesh Window

The finite element mesh window (Fig. 2) displays the generated mesh. In this window, the mesh can be rotated by dragging the mouse, moved in 4 directions by pressing keys of LEFT ARROW, RIGHT ARROW, UP ARROW or DOWN ARROW respectively. The view can be zoomed in (by pressing key 'F9'), out (by pressing key 'F10') or frame (by pressing key 'F11').

To display a 2D view, press key 'F2' (for Plane XY, where X is the longitudinal direction, Y the transverse direction), 'F3' (for Plane YZ, where Z is the vertical direction) or 'F4' (for Plane XZ). An isometric view of the mesh can be achieved by pressing key 'F5'.

Alternatively, users can use the corresponding buttons shown in Fig. 5.

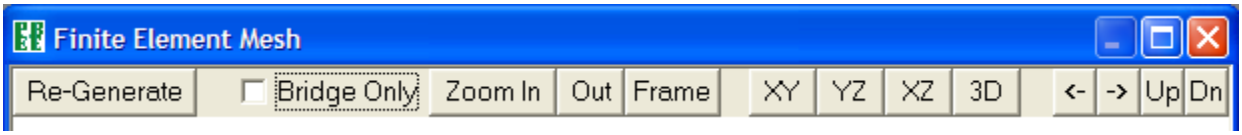


Fig. 5. Buttons available in the Finite Element Mesh window

3 Bridge Model

To define a bridge model, click **Bridge Parameters** in the **Model Input** window (Fig. 6).

Fig. 7 displays the **Bridge Model** window. For a single-bent bridge, essentially four parts are needed to define: column, deck, embankment and abutment.

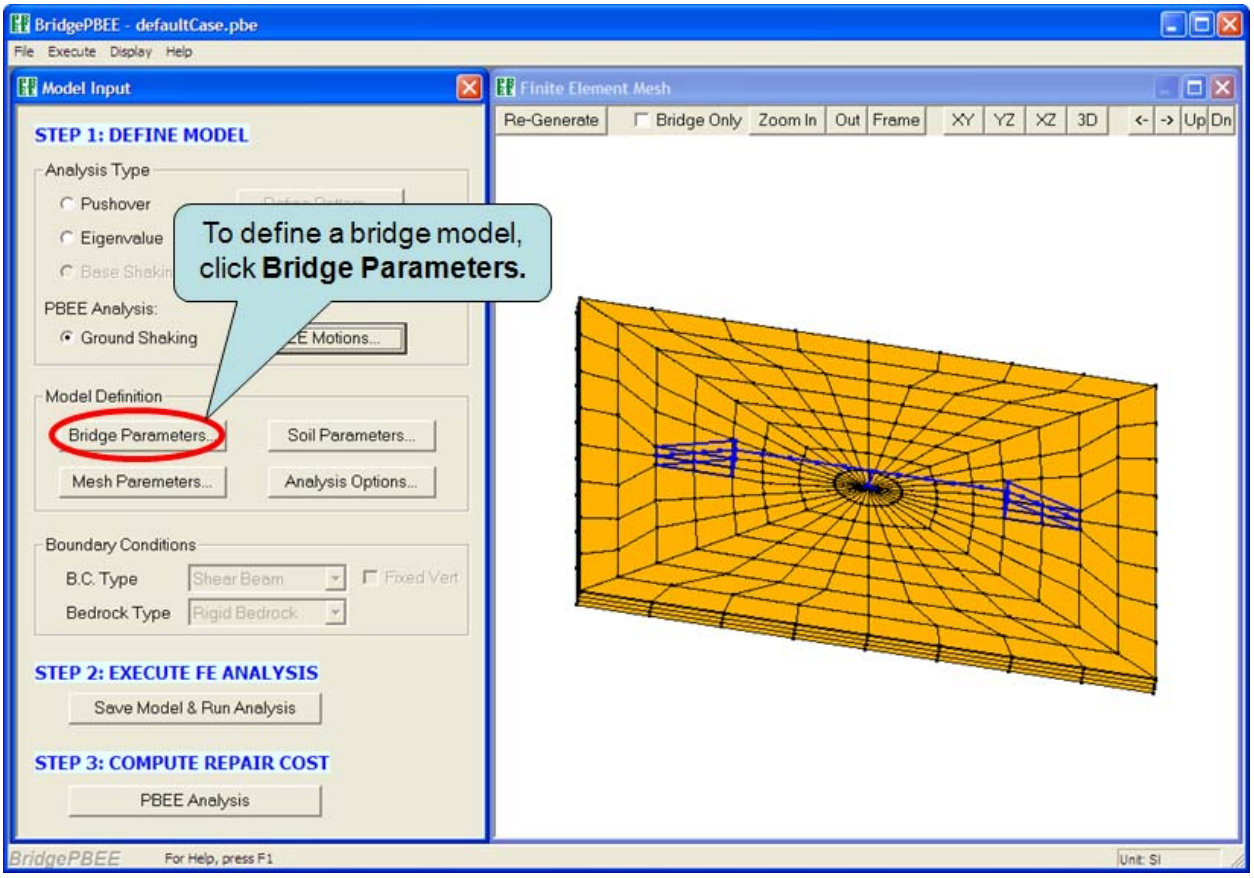


Fig. 6. BridgePBEE main window (defining a bridge model)

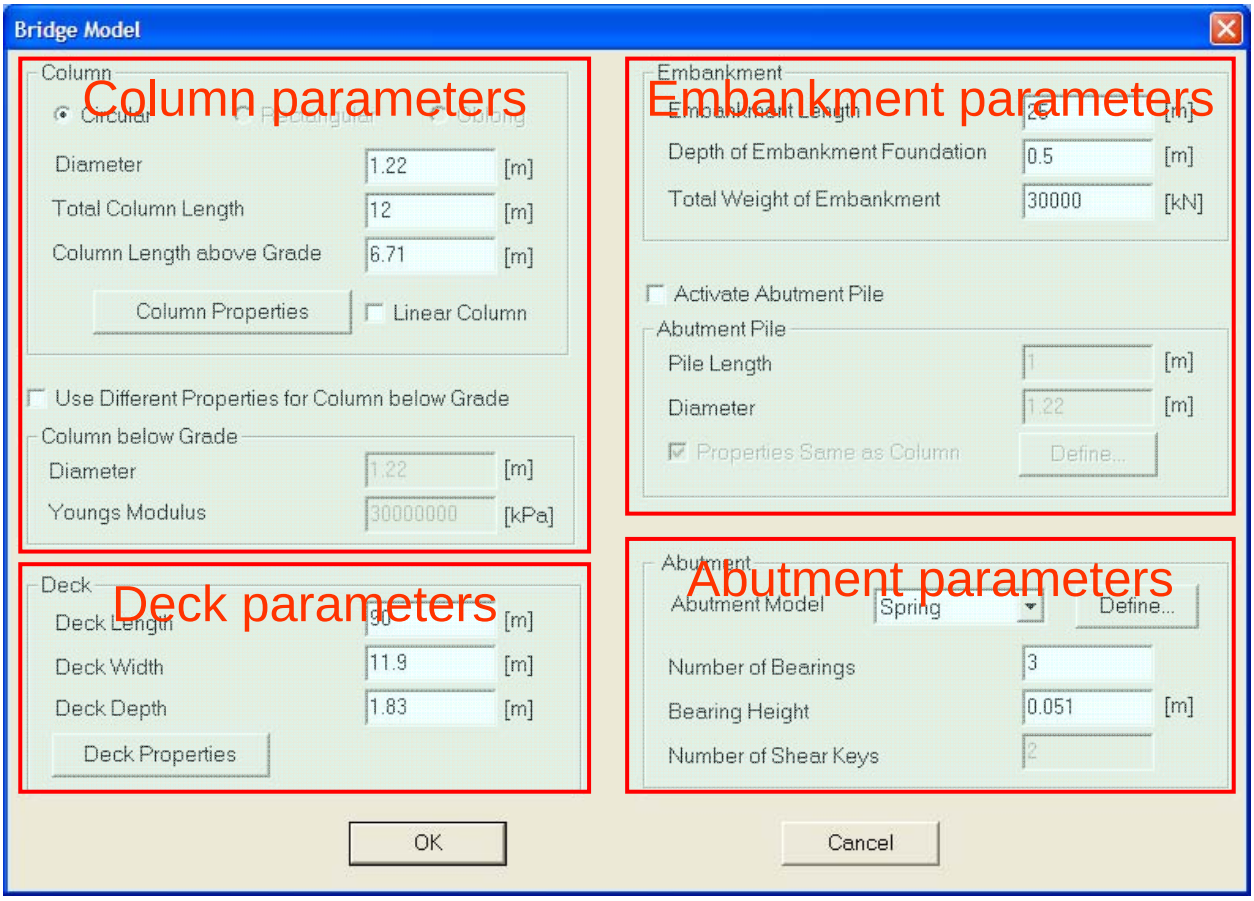


Fig. 7. Bridge Model window

3.1 Column Parameters

Parameters to define the geometrical configurations of the column include (refer to Fig. 7):

- **Circular**: column cross section type. Currently only circular cross section is available.
- **Diameter**: column diameter, which is 1.22 m by default.
- **Total Column Length**: the total length of the column including the pile shaft below grade. The default value is 12 m.
- **Column Length above Grade**: the length of the column above grade. The default value is 6.71 m.

To define the material properties of the column, click **Column Properties**. There are 2 scenarios in this case:

1) Linear material properties will be defined if **Linear Column** is checked.
2) Nonlinear Fiber Section will be defined if **Linear Column** is unchecked.

Please see next section for detailed information.

3.1.1 Column Linear Material Properties

To define the linear material properties of the column, follow the steps shown in Fig. 8. Parameters to define a linear column include (Fig. 8):

- **Young's Modulus**: Young's Modulus of the column. The default value is 3×10^7 kPa.
- **Moment of Inertia @ Transverse Axis**: the default value $I = \pi D^4/64 = 0.108745$ m^4 (where D -- column diameter, D = 1.22 m by default). The default value is for the gross moment of inertia; it can be reduced as desired by the user to better capture cracked column properties.
- **Moment of Inertia @ Longitudinal Axis**: calculation for the default value is the same as the above.
- **Cross-Section Area**: the default value (0.7854 m^2) is calculated based on the circular cross-section with a diameter of 1.22 m.
- **Mass Density**: the mass density of the column. The default value is 2.4 ton/m^3.

Elastic beam-column elements (elasticBeamColumn, Mazzoni et al. 2009) are used for the column in this case.

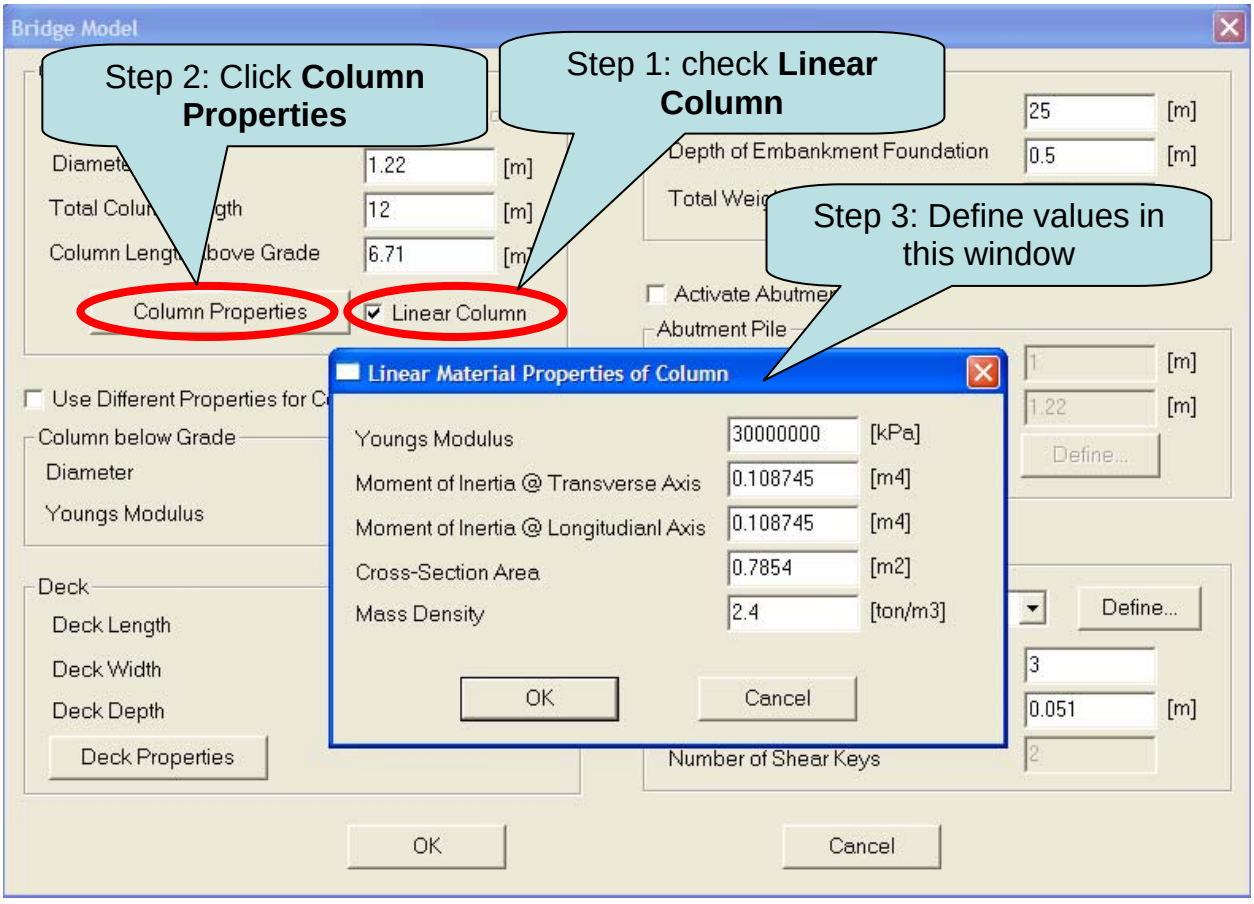


Fig. 8. Steps to define the elastic properties of the column

3.1.2 Nonlinear Fiber Section

To define the nonlinear Fiber section for the column, follow the steps shown in Fig. 9. The window to define the Fiber section is shown in Fig. 10. Nonlinear beam-column elements with fiber section (Fig. 11) are used to simulate the column/pile shaft in this case. Forced-based beam-column elements (nonlinearBeamColumn, Mazzoni et al. 2009) are used for the column (1 element, number of integration points = 5) as well as the pile shaft below grade (number of integration points = 3). The default values for the material properties of the column/pile shaft are shown in Tables 1-3.

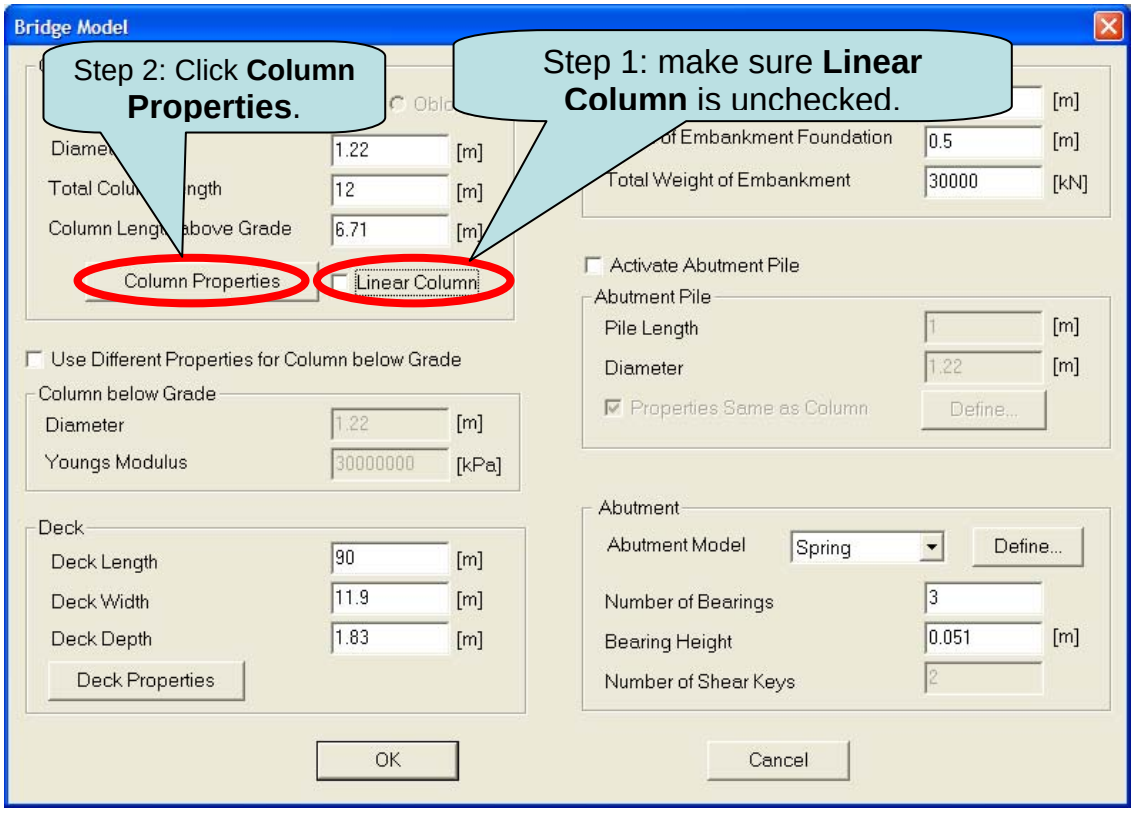


Fig. 9. Steps to define a nonlinear Fiber Section

By default, the Steel02 material in OpenSees (Mazzoni et al. 2009) is employed to simulate the steel bars and Concrete02 material is used for the concrete (core and cover). Steel02 is a uniaxial Giuffré-Menegotto-Pinto material that allows for isotropic strain hardening. Concrete02 is a uniaxial material with linear tension softening. The default values for the material properties of the Fiber section are listed in Table 2 for Steel02 and Table 3 for Concrete02 (core and cover). The Concrete02 material parameters were obtained from the Mander (1988) constitutive relationships for confined and unconfined concrete. More details on the derivation of the default values and the OpenSees uniaxialMaterial definitions used for each material are shown in Appendix D.

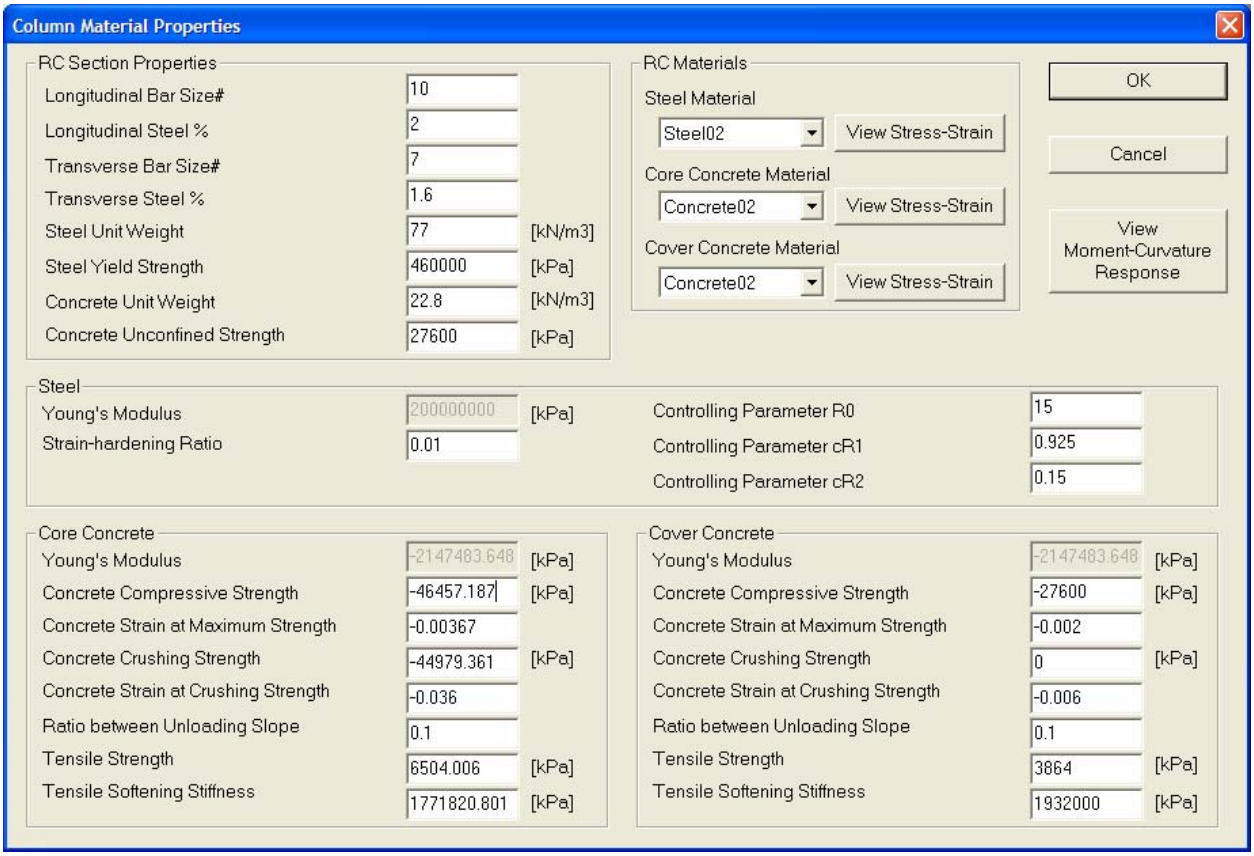


Fig. 10. Fiber Section window

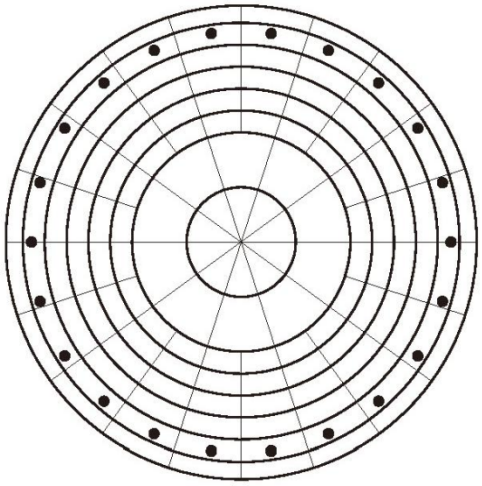


Fig. 11. Column fiber section (based on PEER best modeling practices report, Berry and Eberhard, 2007)

Table 1. Default Values for Column Reinforced Concrete (RC) Section Properties

Parameter	Value
Longitudinal bar size (US #)	10
Longitudinal steel %	2
Transverse bar size (US #)	7
Transverse steel %	1.6
Steel unit weight (kN/m^3)	77
Steel yield strength (kPa)	460,000
Concrete unit weight (kN/m^3)	22.8
Concrete unconfined strength (kPa)	27,600

Table 2. Default Values for Steel02 Material Properties

Parameter	Value	Typical range
Steel yield strength (kPa)	460,000	345,000-470,000
Young's modulus (MPa)	200,000	-
Strain-hardening ratio*	0.01	0.005-0.025
Controlling parameter R0**	15	10-20
Controlling parameter cR1**	0.925	--
Controlling parameter cR2**	0.15	--

*The strain-hardening ratio is the ratio between the post-yield stiffness and the initial elastic stiffness.

**The constants R0, cR1 and cR2 are parameters to control the transition from elastic to plastic branches.

Table 3. Default values for Concrete02 Material Properties

Parameter	Core	Cover
Elastic modulus (MPa)	25,312	25,312
Compressive strength (kPa)	-46,457	-27,600
Strain at maximum strength	-0.00367	-0.002
Crushing strength (kPa)	-44,979	0
Strain at crushing strength	-0.036	-0.006
Ratio between unloading slope	0.1	0.1
Tensile strength (kPa)	6504	3864
Tensile softening stiffness (kPa)	1,771,820	1,932,000

Figs. 12-14 show the stress-strain curves for the steel, core, and cover concrete materials, respectively. These plots can be obtained for updated material properties directly from the interface by clicking on the corresponding View Stress-Strain buttons in the Column Material Properties window (Fig. 10). The moment-curvature response for the column is shown in Fig. 15 (generated with consideration of the overall deck weight 11,915 kN applied at the column top).

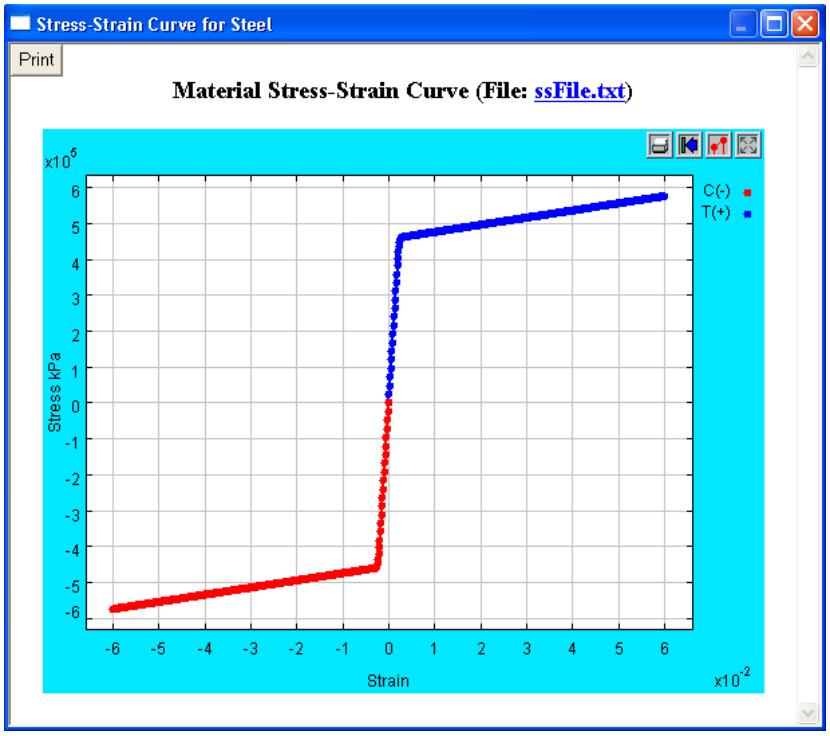


Fig. 12. Stress-strain curve for Steel02 material (default values employed; "C" represents compression and "T" represents tension)

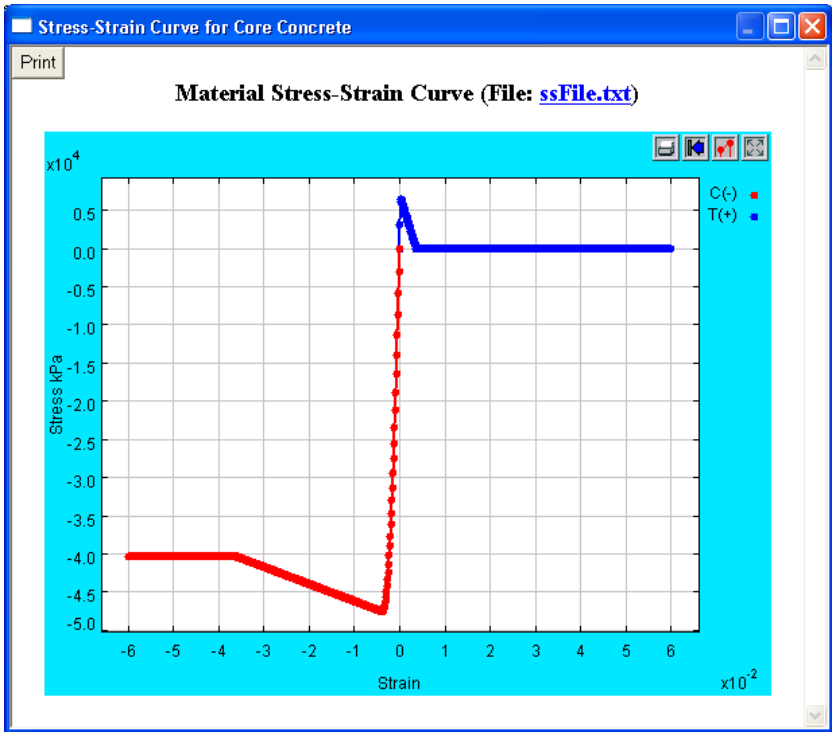


Fig. 13. Stress-strain curve of Concrete02 material for the core concrete (default values employed; "C" represents compression and "T" represents tension)

Important note: The above-displayed graphics applet allows for mouse-driven zoom capability (To zoom, just left-click and drag at the desired location)

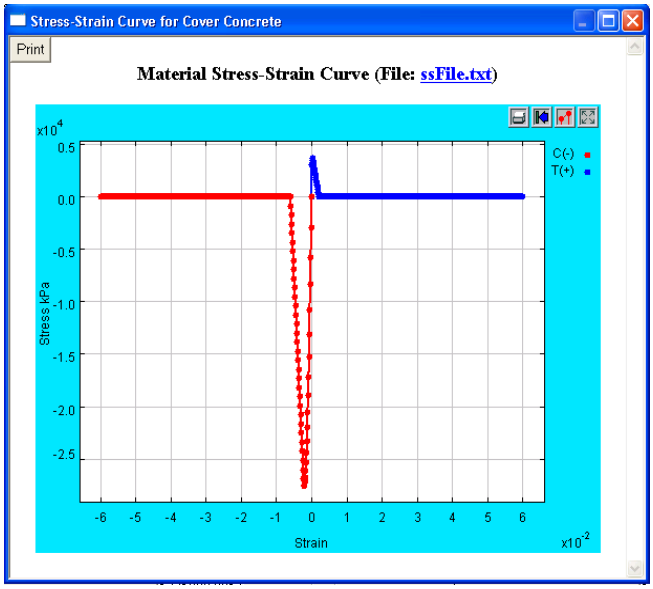


Fig. 14. Stress-strain curve of Concrete02 material for the cover concrete (default values employed; "C" represents compression and "T" represents tension)

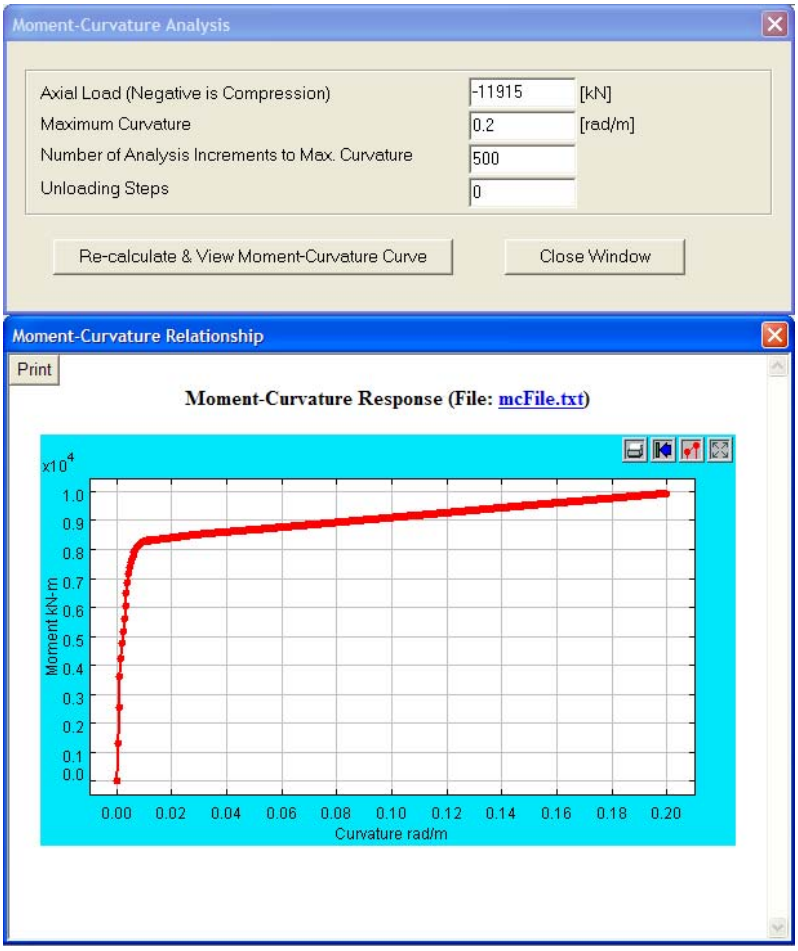


Fig. 15. Moment-curvature response for the column (with default steel and concrete parameters, and the deck weight 11,915 kN applied at the column top)

3.1.3 Column below Grade

If **Use Different Properties for Column below Grade** is checked (Fig. 7), the column below grade can be different from the portion above grade. In this case, the column below grade is assumed to be elastic only. The column diameter and the Young's Modulus are required to define (Fig. 7) the properties of this elastic column below grade.

3.2 Bridge Deck Parameters

To define the deck, please follow the steps shown in Fig. 16. The default values are listed in Tables 4 and 5 below. The default values were obtained from a two-cell reinforced concrete box girder deck configuration.

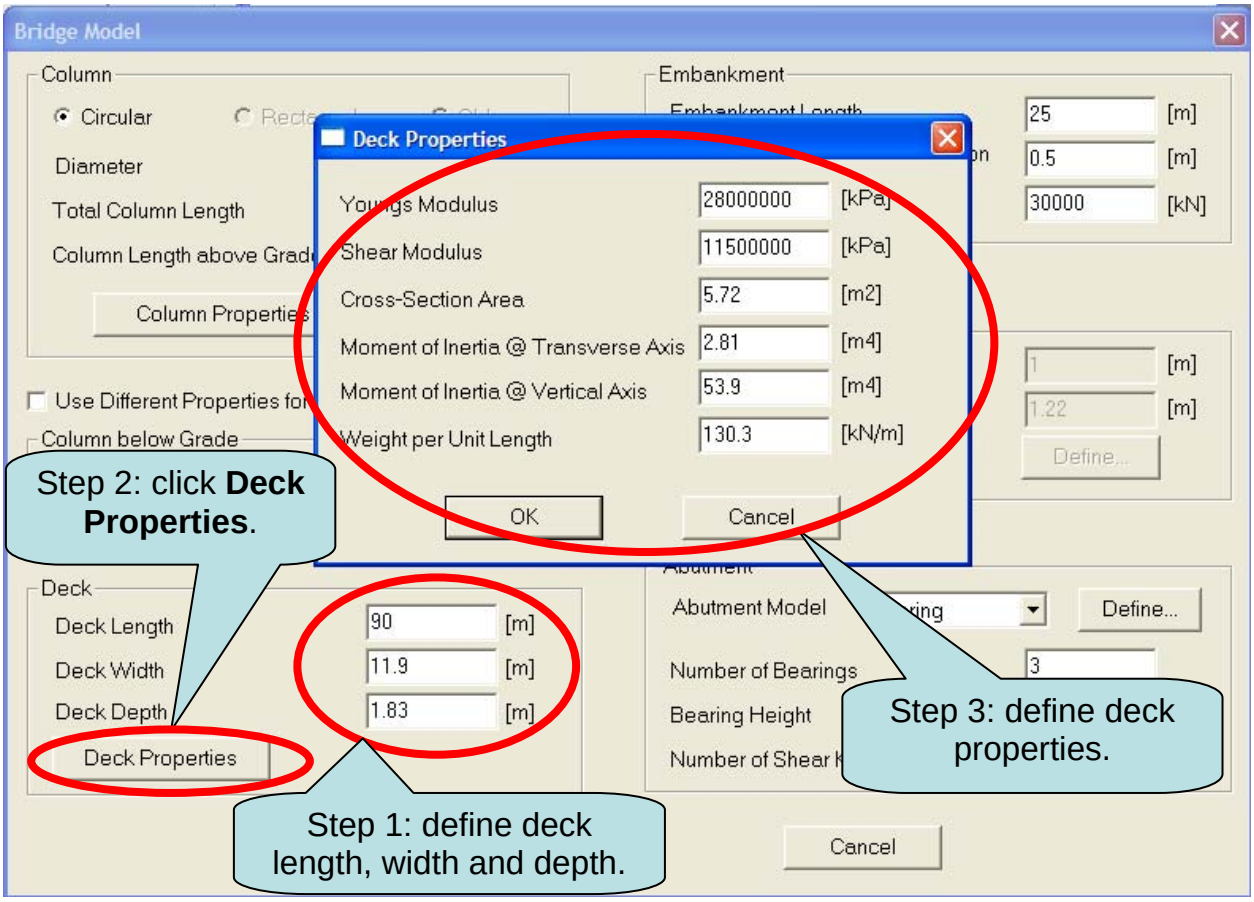


Fig. 16. Steps to define the deck geometrical configuration and material properties

Table 4. Default Values for Bridge Deck

Parameter	Value
Deck length (m)	90.0
Deck width (m)	11.9
Deck depth (m)	1.83

Table 5. Default Values for Deck Material Properties

Parameter	Value
Elastic modulus (MPa)	28,000
Shear modulus (MPa)	11,500
Cross-section area (m^2)	5.72
Moment of inertia @ transverse axis (m^4)	2.81
Moment of inertia @ vertical axis (m^4)	53.9
Weight per unit length (kN/m)	130.3

3.3 Embankment Parameters

The geometric configuration of the embankments is shown in Fig. 2 by the triangular shapes to the right and left of the bridge deck. These geometric triangular configurations are simply represented by relatively rigid beam-column elements. This simple idealization of the embankment allows for (Fig. 7) representation of the distributed self-weight of soil embankment (if present) and a depth of embankment/abutment foundation into the soil mesh. The user will specify the embankment length in the longitudinal bridge direction, depth of the embankment below grade, and total weight of the embankment (the user must calculate this parameter to match the mass of the actual soil embankment). The embankment parameters will have no effect in the rigid ground simulation cases, but will contribute when the bridge is supported on soil mesh.

In addition, a single pile represented only by beam-column elements (i.e., cross-sectional geometry of the pile is not represented) may be included by the user (Fig. 7) to further support the embankment/abutment. The single pile is positioned below the embankment geometric configuration (closest to the bridge, and aligned with the bridge longitudinal axis). This option is activated by selecting the checkbox **Activate Abutment Pile.** Length of this additional pile can be specified as well as its diameter. The material properties of this pile can be the same as the bridge's central column, or can be defined independently by clicking "**Define**" as shown in Fig. 7. Upon clicking define, a window similar to that of Fig. 10 will open and the user can follow the procedures associated with Fig. 10 as described earlier.

3.4 Abutment Parameters

Abutment behavior, soil-structure interaction, and embankment flexibility have been found by post-earthquake reconnaissance reports to significantly influence the response of the entire bridge system under moderate to strong intensity ground motions. Specifically, for Ordinary Standard bridge structures in California with short spans and relatively high superstructure stiffness, the embankment mobilization and the inelastic behavior of the soil material under high shear deformation levels dominate the response of the bridge and the intermediate column bents (Kotsoglu and Pantazopoulou, 2006, and Shamsabadi et al. 2007, 2010). Seven abutment models are implemented in BridgePBEE. The abutment models are defined as Elastic, Roller, Simplified (SDC 2004), Spring (SDC 2004), SDC 2010 Sand, SDC 2010 Clay, and EPP-Gap abutment models.

3.4.1 Elastic Abutment

The elastic abutment model (Fig. 17) consists of a simple set of 6 translational elastic springs at each end of the bridge (see schematic below): 2 longitudinal, 2 transverse, and 2 vertical springs. By default, no additional rotational springs are specified, but can be added by the user.

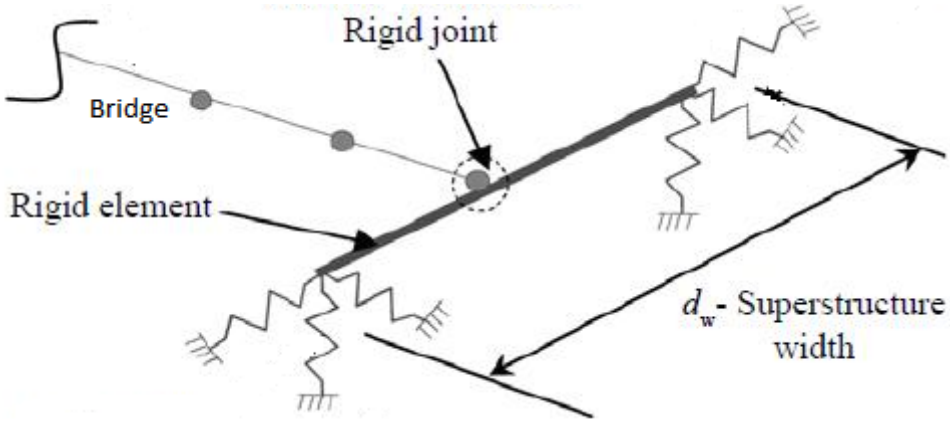


Fig. 17. Elastic abutment model

To define an Elastic abutment model, please follow the steps shown in Fig. 18. The typical force-displacement curve for the Elastic abutment model is shown in Fig. 19.

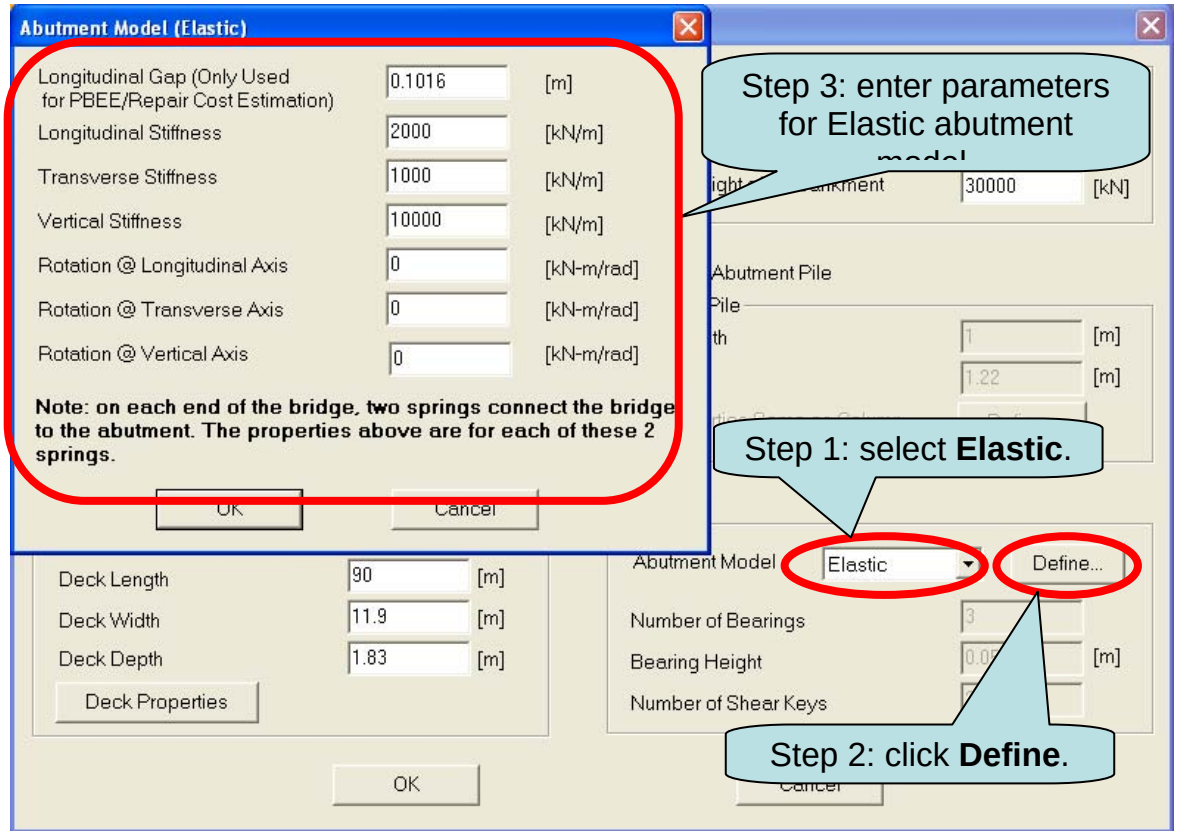


Fig. 18. Steps to define Elastic abutment model

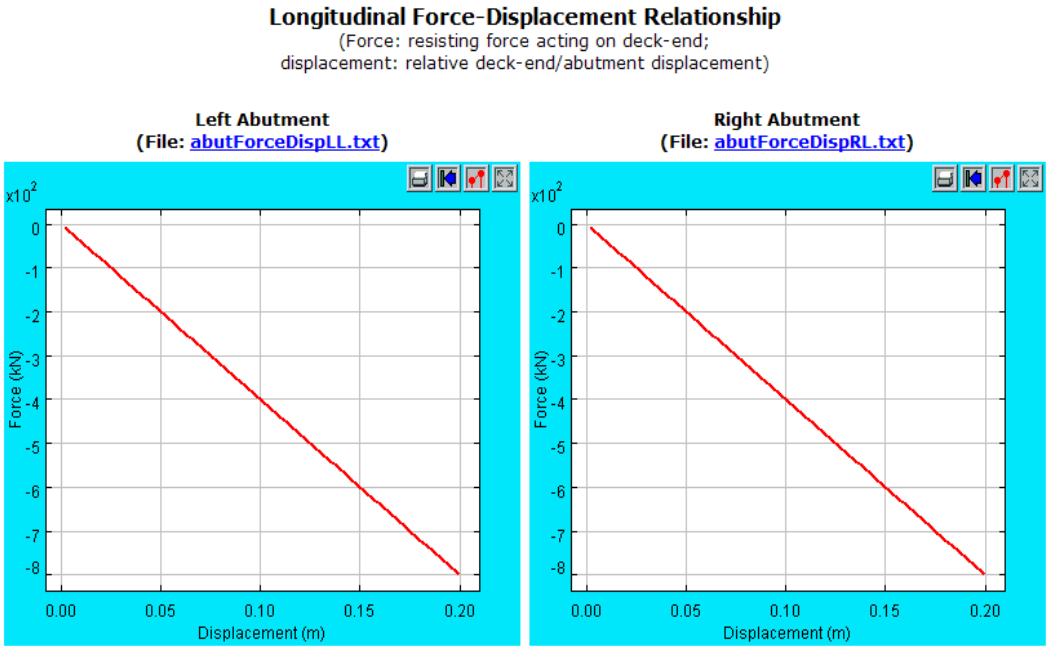


Fig. 19. Longitudinal force-displacement relationship for the Elastic abutment model

3.4.2 Roller Model

The roller abutment model (Fig. 20) consists of rollers in the transverse and longitudinal directions, and a simple boundary condition module that applies single-point constraints against displacement in the vertical direction (i.e., bridge and abutment are rigidly connected in the vertical direction). These vertical restraints also provide a boundary that prevents rotation of the deck about its axis (torsion).

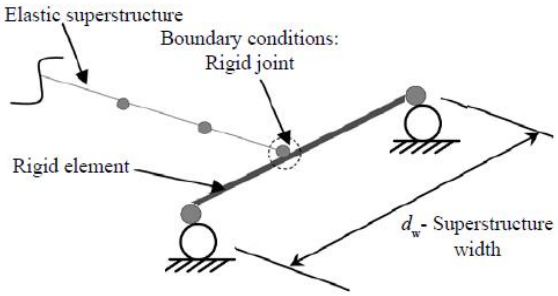


Fig. 20. Roller abutment model

This model can be used to provide a lower-bound estimate of the longitudinal and transverse resistance of the bridge, that may be displayed through a pushover analysis. To define a Roller abutment model, please follow the steps shown in Fig. 21. The typical force-displacement curve for the Roller abutment model is shown in Fig. 22.

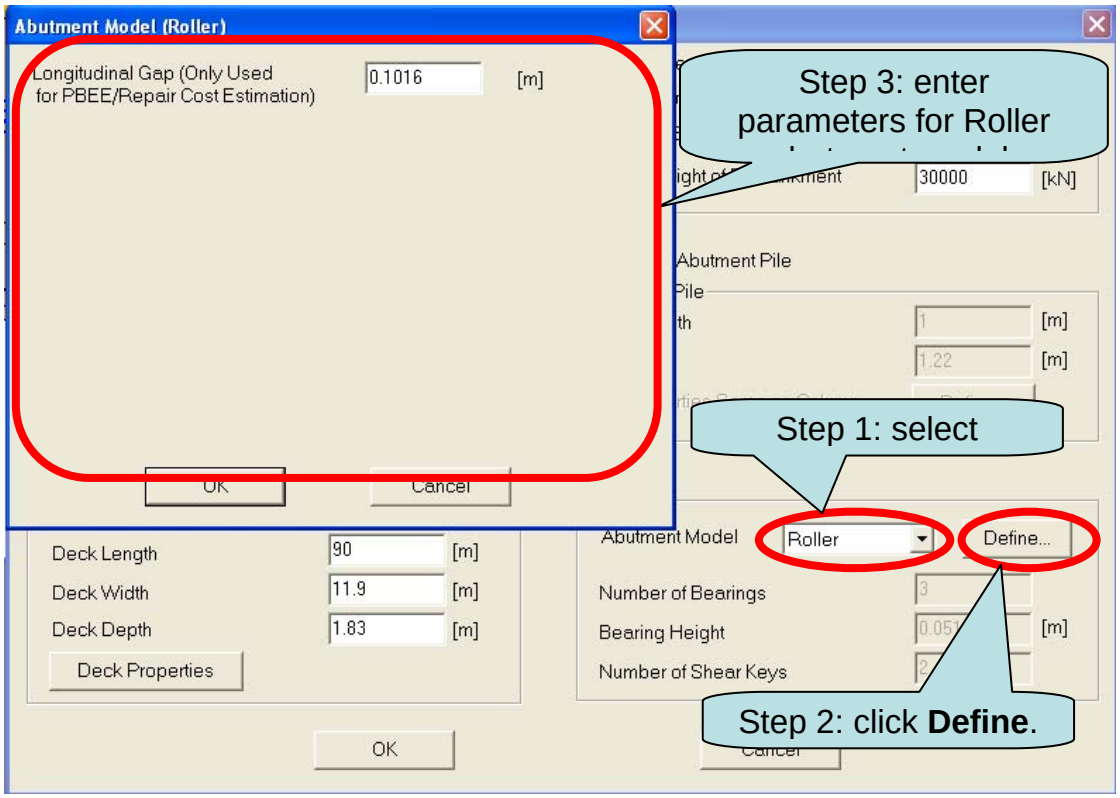


Fig. 21. Steps to define a Roller abutment model

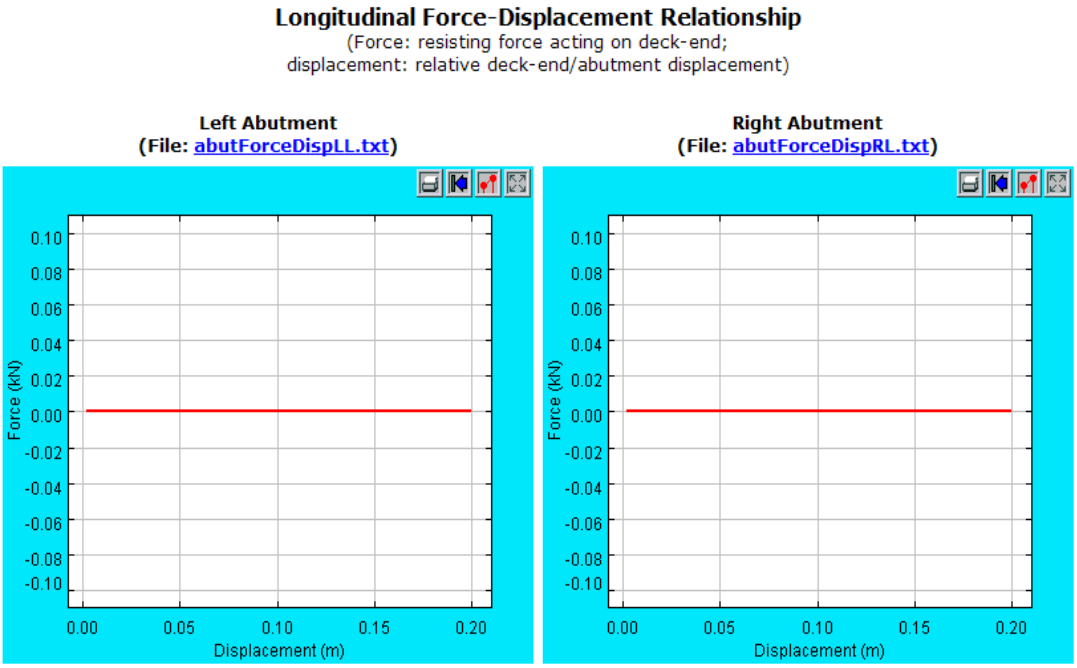


Fig. 22. Longitudinal force-displacement relationship for the Roller abutment model

3.4.3 Simplified Model (SDC 2004)

The simplified model of the embankment-abutment system provides several nonlinear springs to better represent abutment-bridge interaction that is neglected with the elastic or roller abutment models. The general scheme of the simplified model is presented in Fig. 23. It consists of a rigid element of length d_w (superstructure width), connected through a rigid joint to the superstructure centerline, with defined longitudinal, transverse and vertical nonlinear response at each end.

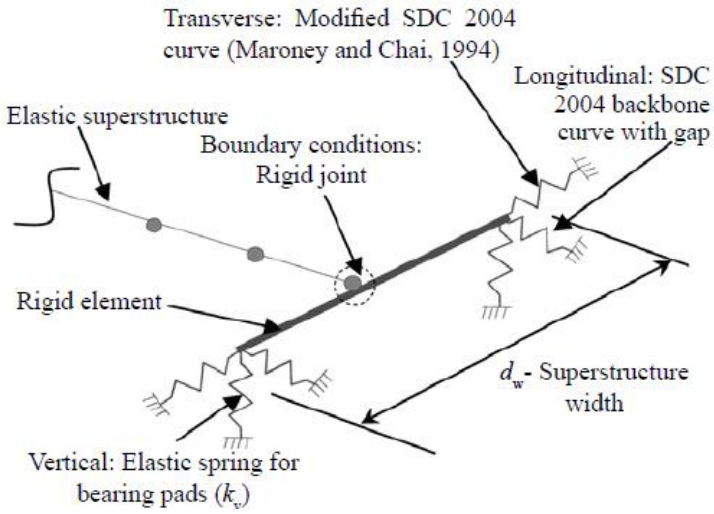

Transverse: Modified SDC 2004 curve (Maroney and Chai, 1994)
Longitudinal: SDC 2004 backbone curve with gap
Elastic superstructure
Boundary conditions: Rigid joint
Rigid element
d_w - Superstructure width
Vertical: Elastic spring for bearing pads (k_v)

Fig. 23. General scheme of the Simplified abutment model (Aviram *et al.*, 2008)

The longitudinal response defined for the simplified model accounts only for the gap and the embankment fill response, where passive pressures are produced by the abutment back wall (Fig. 23). The shear resistance of bearing pads connecting the bridge to the abutment wall is ignored. In the longitudinal direction (Fig. 23), a gap element is assigned an elastic-perfectly-plastic (EPP) backbone curve after gap closure with abutment stiffness (K_{abut}) and ultimate strength (P_{abut}) obtained from section 7.8.1 of the Caltrans SDC (2004), see Fig. 24. There is no stiffness in the longitudinal direction when the deck pulls away from the abutment.

The stiffness and strength are calculated using the SDC equations:

$$K_{abut} = 11500.0w\left(\frac{h}{1.7}\right) \tag{1}$$

$$P_{abut} = 239.0wh\left(\frac{h}{1.7}\right) \tag{2}$$

Where w is the width of the back wall (unit: m) and h is the height of the back wall (unit: m). In the current implementation, the width of the back wall is taken as the bridge deck width minus twice of the bridge deck depth. The units of K_{abut} and P_{abut} are kN/m and kN, respectively.

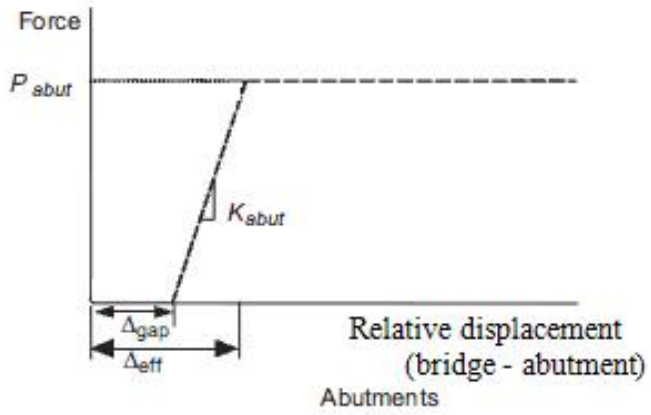


Fig. 24. Longitudinal backbone curve force-displacement relationship (two on each end of the bridge; Caltrans SDC, 2004)

In the transverse direction, a zero-length element is defined at each end of the rigid link with an assigned Elastic perfectly plastic (EPP) backbone curve representing the wing wall and pile resistance, similar to the longitudinal backbone. The transverse backbone is obtained by multiplying the longitudinal backbone by C_L = 2/3 and C_W = 4/3 and is mobilized immediately (there is no gap in the transverse direction). The resistance of the brittle shear keys and distributed bearing pads is ignored in this model for simplicity. Skew changes the orientation of the rigid link at the end of the deck segment.

In the vertical direction, an elastic spring is defined at each end of the rigid link, with a stiffness corresponding to the vertical stiffness of the embankment soil mass. The embankment is assumed to have a trapezoidal shape and based on the effective length formulas from Zhang and Makris (2002), the vertical stiffness (K_v, unit: 1/m) can be calculated from (Zhang and Makris, 2002):

$$K_v = \frac{E_{sl} d_w}{z_0 \ln\left(\frac{z_0 + H}{z_0}\right)} \tag{3}$$

Where H is the embankment height, d_w is the deck width, $z_0 = 0.5 d_w S$, S is the embankment slope (parameter in window, see Fig. 20), $E_{sl} = 2.8G$, $G = \rho V_s^2$, ρ and V_s are the mass density and the shear wave velocity of the embankment soil, respectively (parameters in window, see Fig. 25).

To define a Simplified abutment model, please specify the parameters shown in Fig. 25. The typical force-displacement curve for the Simplified abutment model is shown in Fig. 26.

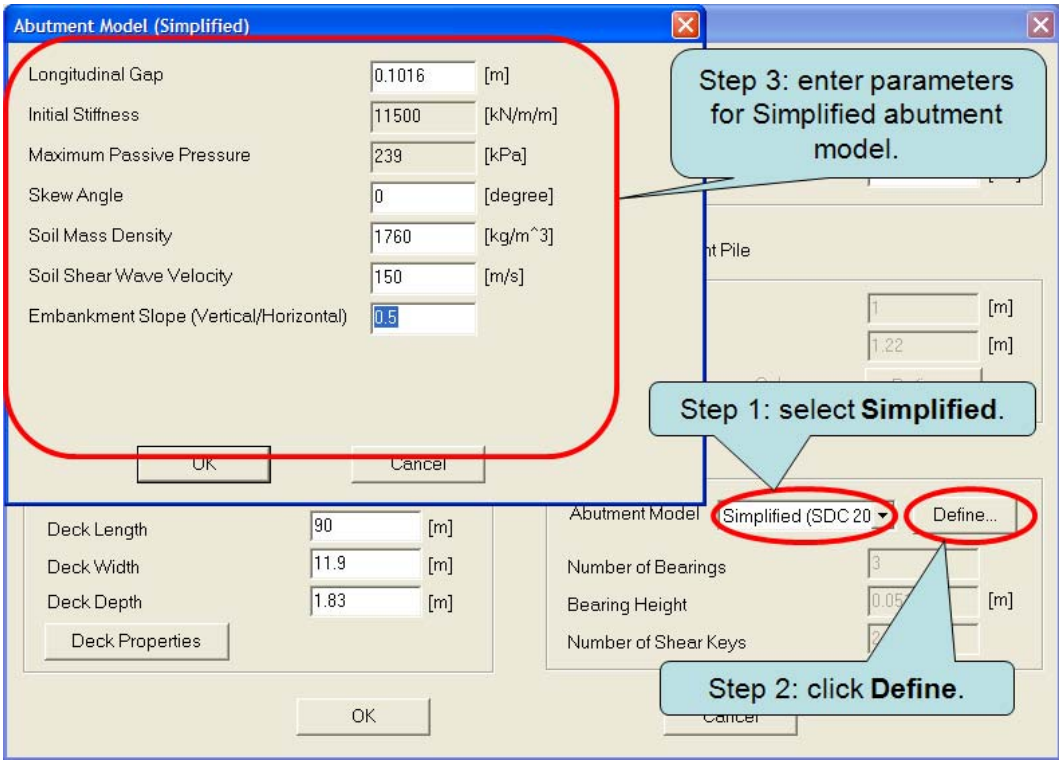
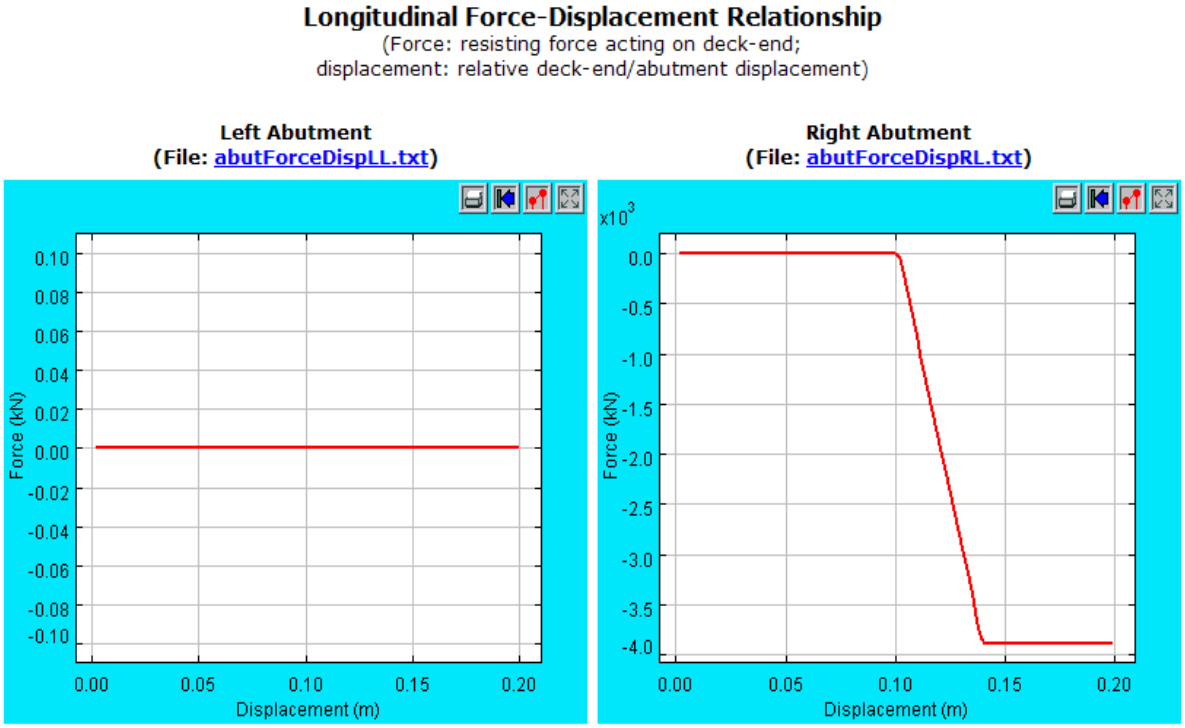
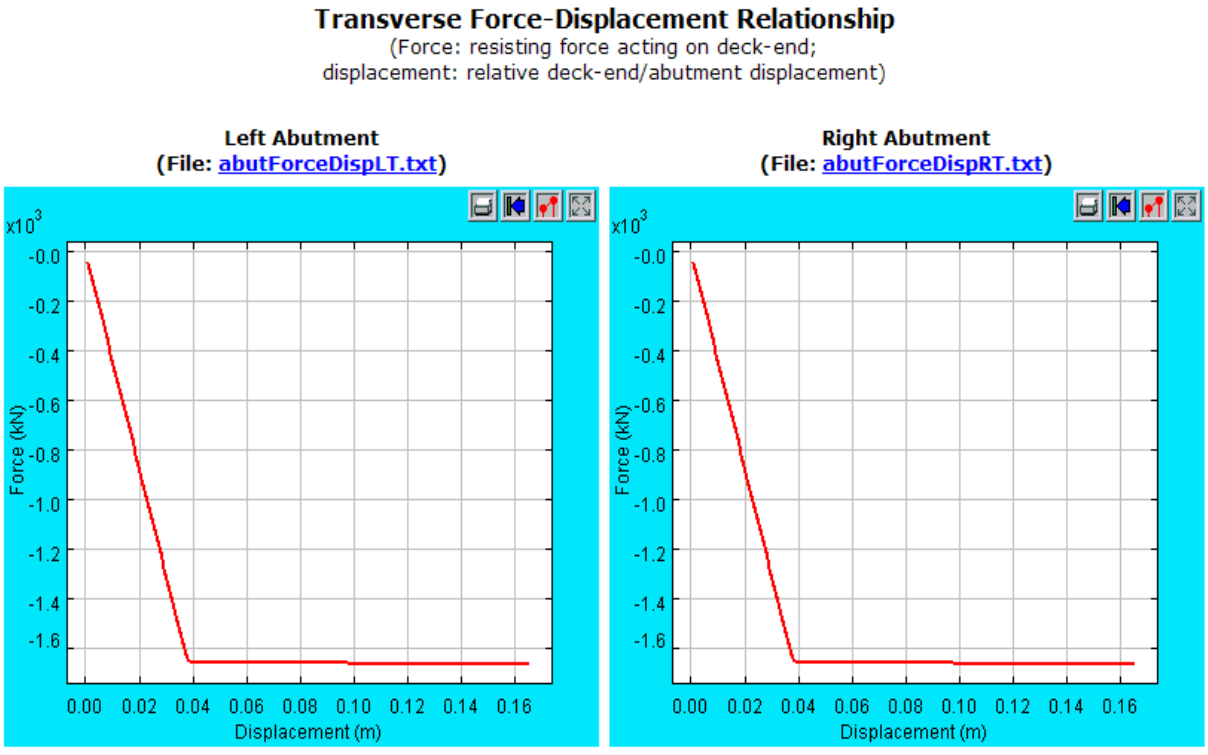


Fig. 25. Steps to define the Simplified abutment model.



a)



b)

Fig. 26. Longitudinal force-displacement relationship for the Simplified abutment model: a) longitudinal direction; b) transverse direction

3.4.4 Spring Model

A more complex abutment model was developed by Mackie and Stojadinovic (2006), including sophisticated longitudinal, transverse, and vertical nonlinear abutment response, as well as a participating mass corresponding to the concrete abutment and mobilized embankment soil. A system of zero-length elements is distributed along two rigid elements oriented in the transverse bridge direction. The discrete zero-length elements represent each component of the abutment that contributes to the combined behavior and allow for differential response in each element as the superstructure rotates about the vertical bridge axis. A general scheme of this abutment model is presented in Fig. 27. The bearing pads create a series system between the two transverse rigid elements (Rigid element 1 and 2 in Fig. 27). Rigid element 1 is connected to the deck end by a rigid joint. The longitudinal elastomeric bearing pad response and gap closure behavior are illustrated by L1 in Fig. 27. The number and distribution of the bearing pads is defined according to the number and location of the girders in the box, with plan and thickness dimensions according to plans or specifications. The longitudinal backfill, back wall, and pile system response are accounted for by the two zero-length elements at the extreme locations of rigid element 2, designated as L2.

Longitudinal response: The longitudinal response is based on the system response of the elastomeric bearing pads, gap, abutment back wall, abutment piles, and soil backfill material. Prior to impact or gap closure, the superstructure forces are transmitted through the elastomeric bearing pads to the stem wall, and subsequently to the piles and backfill, in a series system. After gap closure, the superstructure bears directly on the abutment back wall and mobilizes the full passive backfill pressure.

Transverse Response: The transverse response is based on the system response of the elastomeric bearing pads, exterior concrete shear keys, abutment piles, wing walls, and backfill material. The bearing pad model discussed above is used with uncoupled behavior with respect to the longitudinal direction. The constitutive model of the exterior shear keys is derived from experimental tests (Megally et al., 2003). The parallel system of transverse bearing pads and shear keys are labeled T1 in Fig. 27.

Vertical Response: The vertical response of the abutment model includes the vertical stiffness of the bearing pads (V1) in series with the vertical stiffness of the trapezoidal embankment (V2). The user can modify the vertical tensile force factor for the bearing pads (multiplier on the vertical bearing strength). The embankment stiffness per unit length of embankment was obtained from Zhang and Makris (2000) and modified using the critical length to obtain a lumped stiffness.

Model Characteristics

Each bearing pad has a height (h) of 0.0508 m (2 in) and a side length (square) of 0.508 m (20 in). The properties of a bearing pad are listed in Table 6.

The abutment is assumed to have a nominal mass proportional to the superstructure dead load at the abutment, including a contribution from structural concrete as well as the participating soil

mass. An average of the embankment lengths obtained from Zhang and Makris (2002) and Werner (1994) is included in the calculation of the participating mass due to the embankment of the abutment. The user can modify the lumped mass through the soil mass.

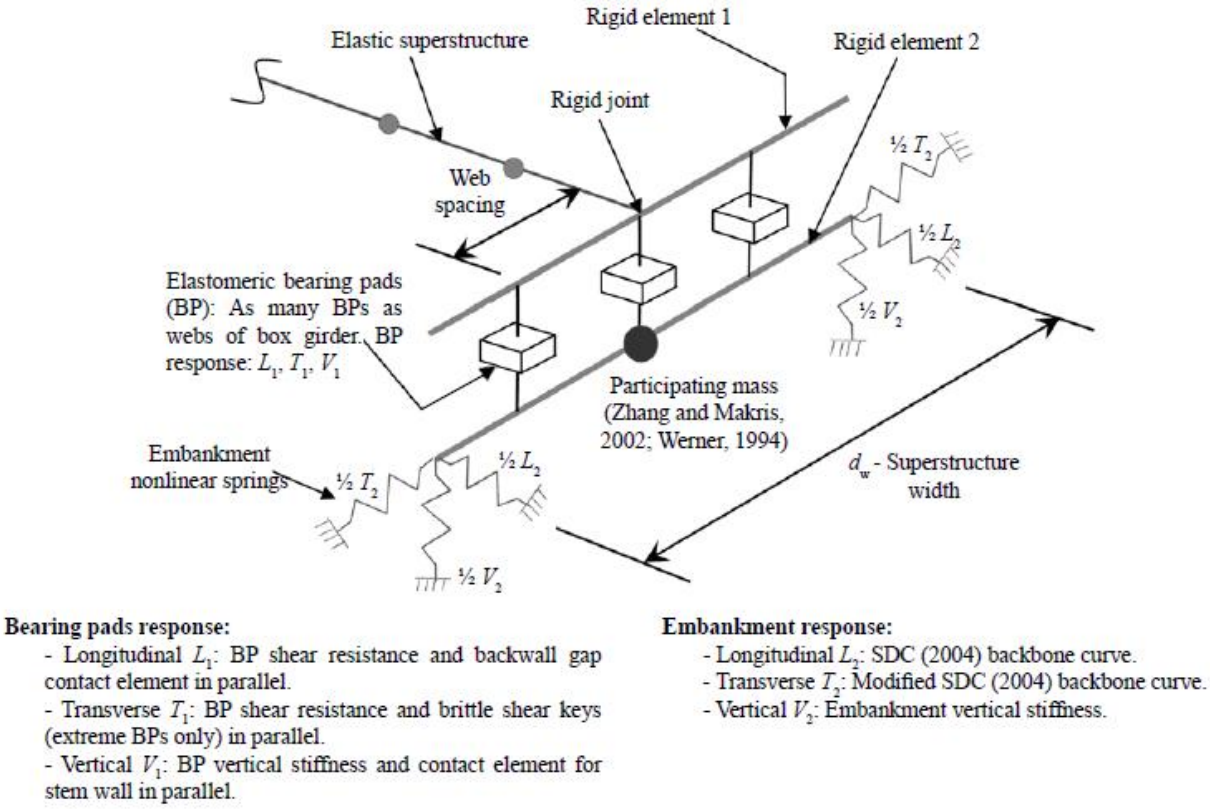

Rigid element 1
Elastic superstructure
Rigid element 2
Rigid joint
½ T_2
Web spacing
½ L_2
Elastomeric bearing pads (BP): As many BPs as webs of box girder. BP response: L_1, T_1, V_1
½ V_2
Participating mass (Zhang and Makris, 2002; Werner, 1994)
Embankment nonlinear springs
½ T_2
½ L_2
d_w - Superstructure width
½ V_2
Bearing pads response:
- Longitudinal L_1: BP shear resistance and backwall gap contact element in parallel.
- Transverse T_1: BP shear resistance and brittle shear keys (extreme BPs only) in parallel.
- Vertical V_1: BP vertical stiffness and contact element for stem wall in parallel.
Embankment response:
- Longitudinal L_2: SDC (2004) backbone curve.
- Transverse T_2: Modified SDC (2004) backbone curve.
- Vertical V_2: Embankment vertical stiffness.

Fig. 27. General scheme of the Spring abutment model (Aviram et al. 2008)

Table 6. Geometric and Material Properties of a Bearing Pad

Shear Modulus G	1034.2 kPa (0.15 ksi)
Young's Modulus E	34473.8 kPa (5 ksi)
Yield Displacement	150% shear strain
Lateral Stiffness	$\frac{GA}{h}$ (where A is the cross-section area and h is the height)
Vertical Stiffness	$\frac{EA}{h}$
Vertical Tearing Stress	15513.2 kPa (2.25 ksi)

To define a Spring abutment model, please follow the steps shown in Fig. 28. The default values for the Spring abutment model are shown in Tables 7 & 8. The typical force-displacement curve for the Spring abutment model is shown in Fig. 29.

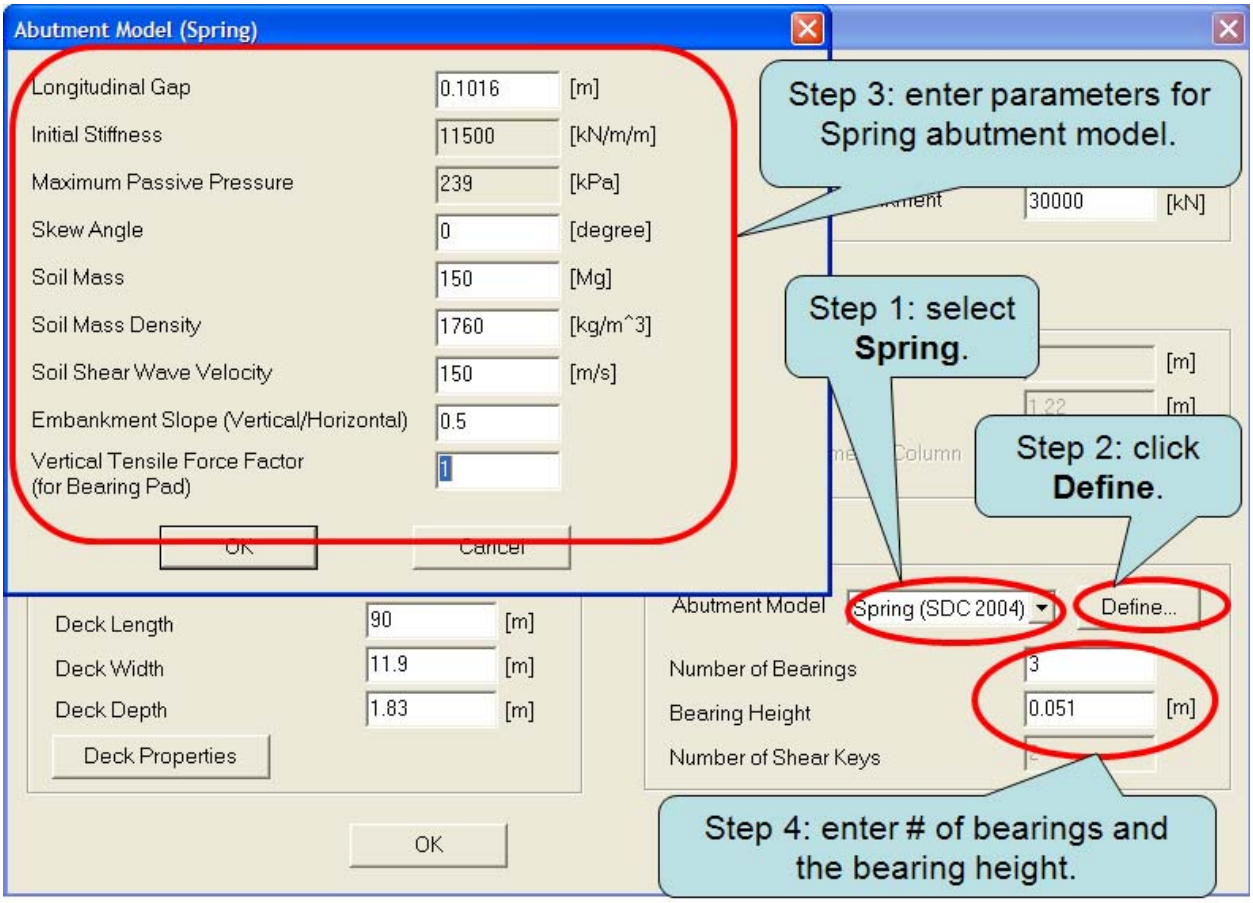


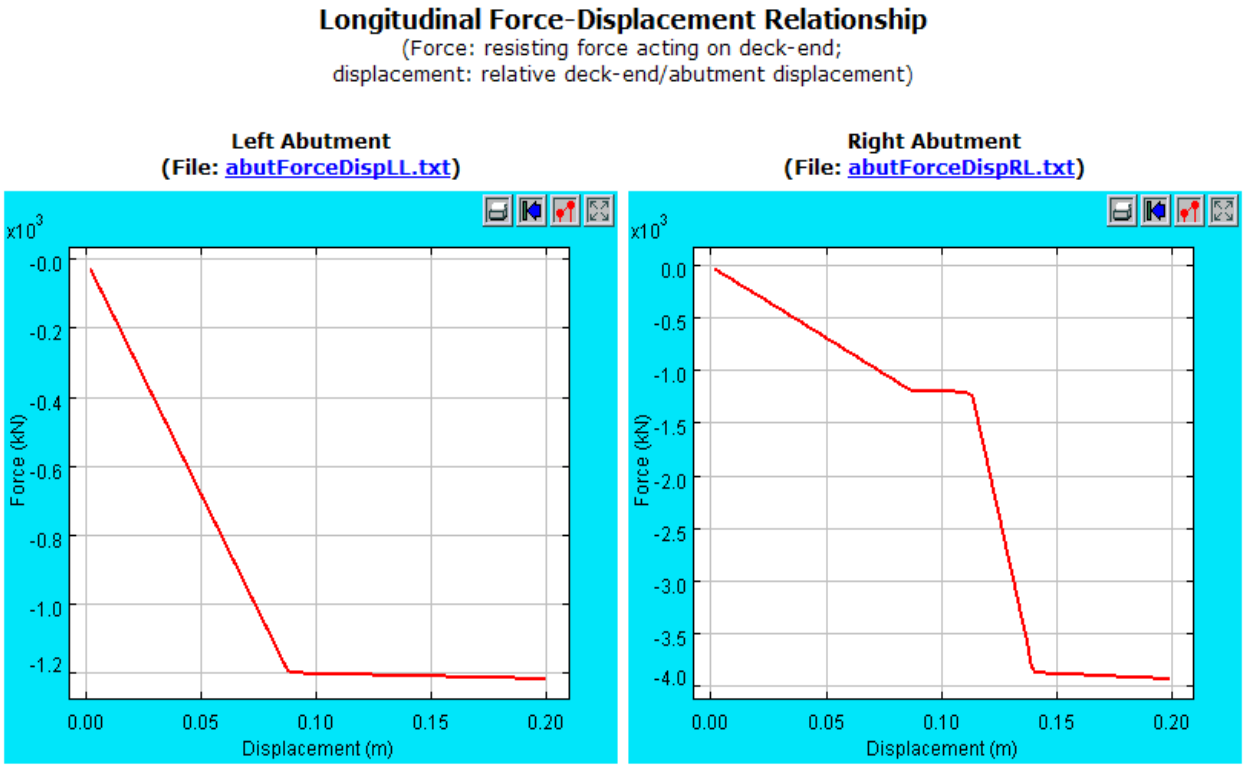
Fig. 28. Steps to define a Spring abutment model

Table 7. Spring Abutment Model Properties

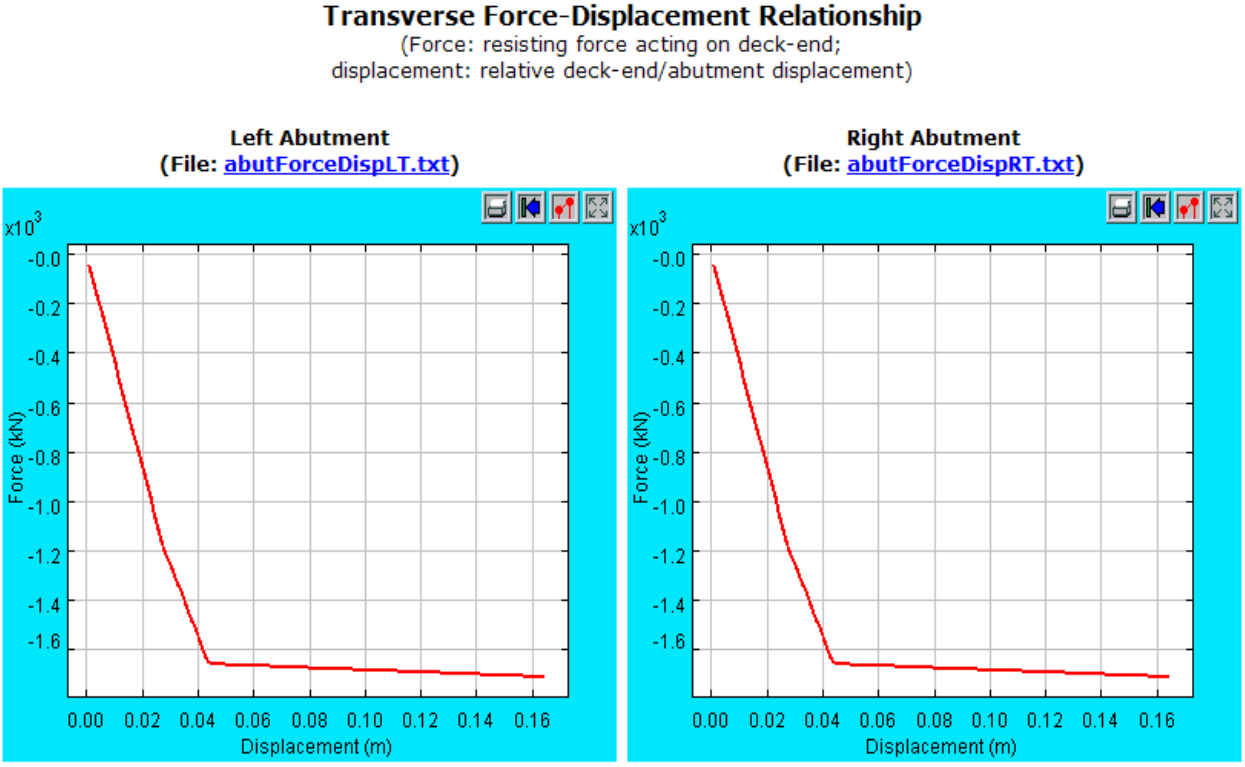
Parameter	Value
Soil mass (Mg)	150
Skew angle (degree)	0
Soil shear wave velocity (m/s)	150
Embankment slope	2
Soil mass density (kg/m^3)	1,760
Longitudinal gap (m)	0.1016

Table 8. Abutment Configurations

Parameter	Value
Number of bearings	3
Bearing height (m)	0.051
Number of shear keys	2
Shear key height (m)	1.83



a)



b)

Fig. 29. Force-displacement relationship for the Spring abutment model: a) longitudinal direction; b) transverse direction.

3.4.5 SDC (2010) Sand

This model is similar to the Simplified (SDC 2004) abutment model, but employs the parameters of the most recent SDC (2010) for a sand backfill Embankment. To define a SDC 2010 Sand abutment model, please follow the steps shown in Fig. 30.

3.4.6 SDC (2010) Clay

This model is similar to the Simplified (SDC 2004) abutment model, but employs the parameters of the most recent SDC (2010) for a Clay backfill Embankment. To define a SDC 2010 Clay abutment model, please follow the steps shown in Fig. 31.

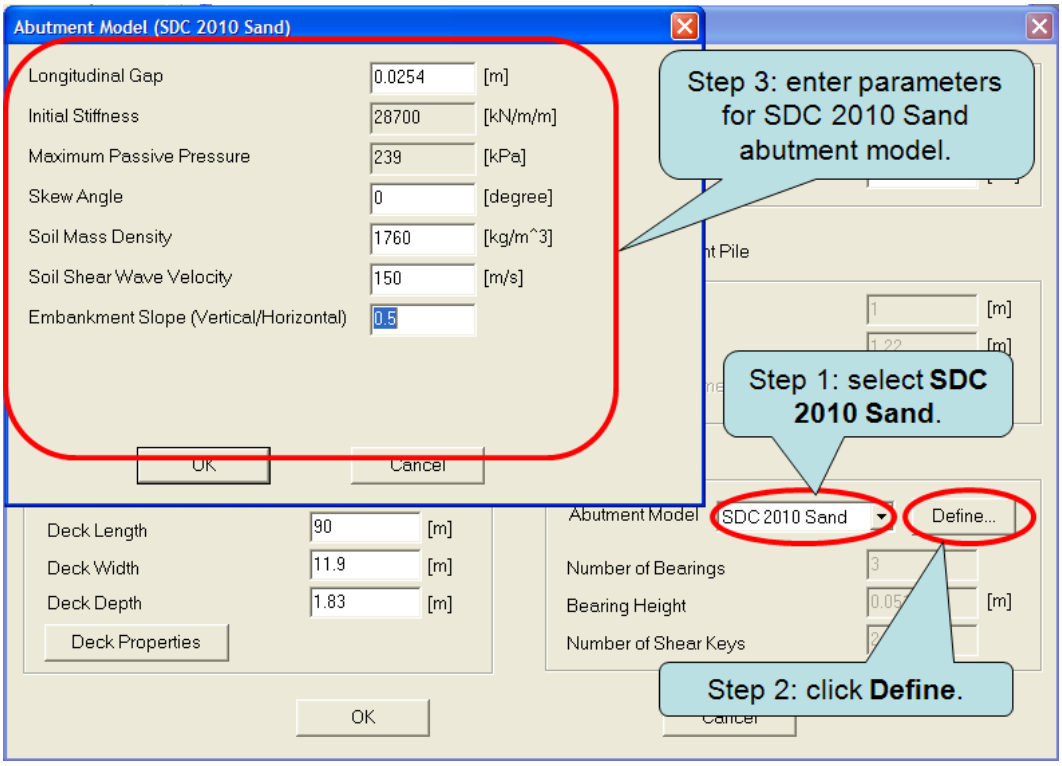


Fig. 30. Steps to define a SDC 2010 Sand abutment model

3.4.7 EPP-Gap Model

This model is similar to the Simplified (SDC 2004) abutment model, but employs user defined parameters for the stiffness, maximum resistance, and gap size between bridge-deck and back-wall. To define an EPP-Gap abutment model, please follow the steps shown in Fig. 32.

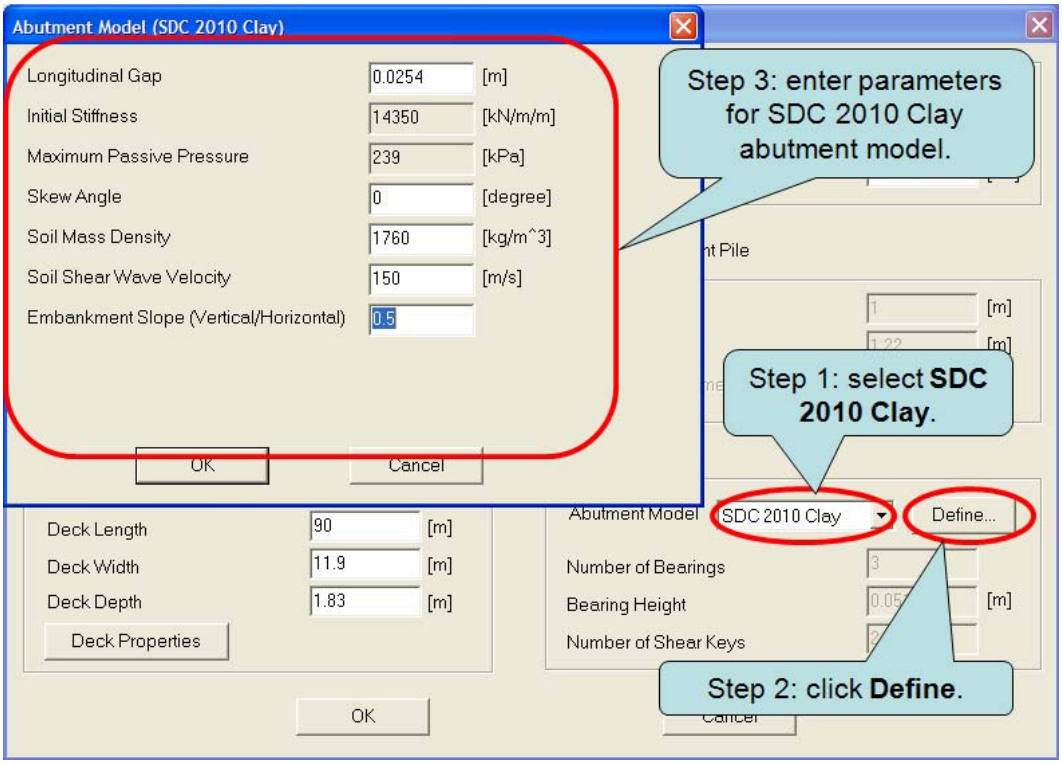


Fig. 31. Steps to define a SDC 2010 Clay abutment model

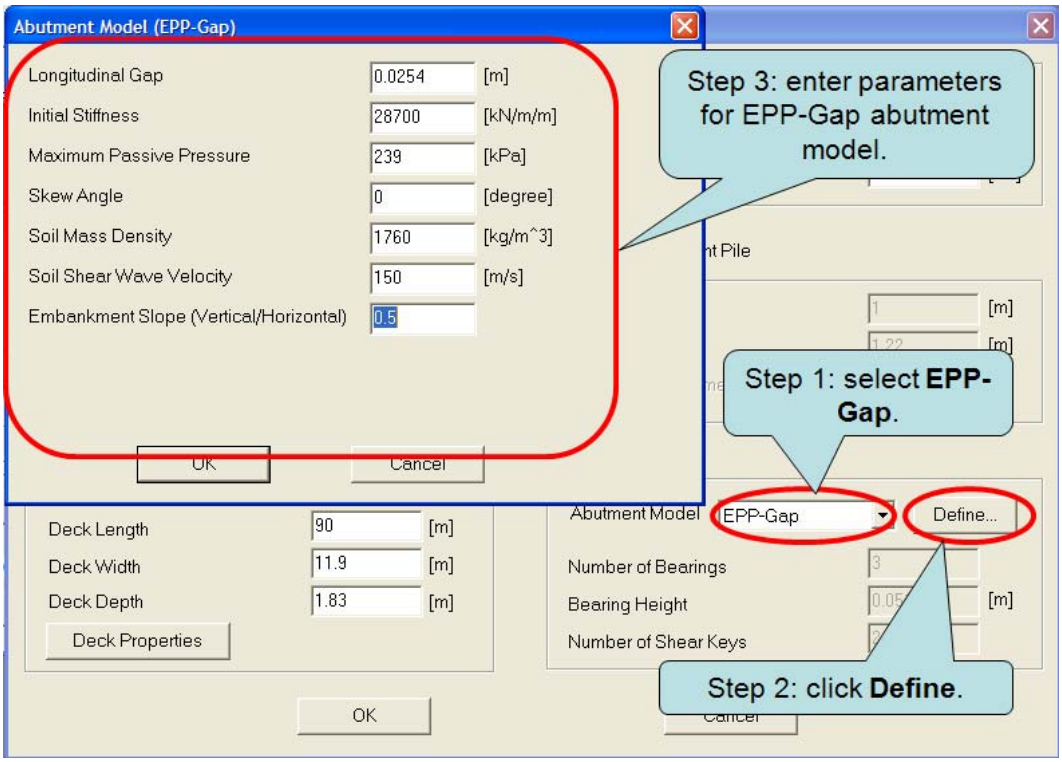


Fig. 32. Steps to define an EPP-Gap abutment model

3.4.8 HFD Model

As suggested by Shamsabadi et al. (2007, 2010), a Hyperbolic Force-Displacement (HFD) relationship is employed to represent abutment resistance to bridge displacement in the longitudinal direction (Fig. 33). In this HFD model, resistance appears after a user-specified gap is traversed (Fig. 33b), and the bridge thereafter gradually mobilizes the abutment's passive earth pressure strength. Herein, this strength is specified according to Shamsabadi et al. (2007, 2010) at 265 kPa (for a nominal 1.7 m bridge deck height), with full resistance occurring at a passive lateral displacement of 0.09 m (the sand structural backfill scenario). Similarly, abutment resistance to the transverse bridge displacement is derived from the longitudinal hyperbolic force-displacement relationship according to the procedure outlined in Aviram et al. (2008).

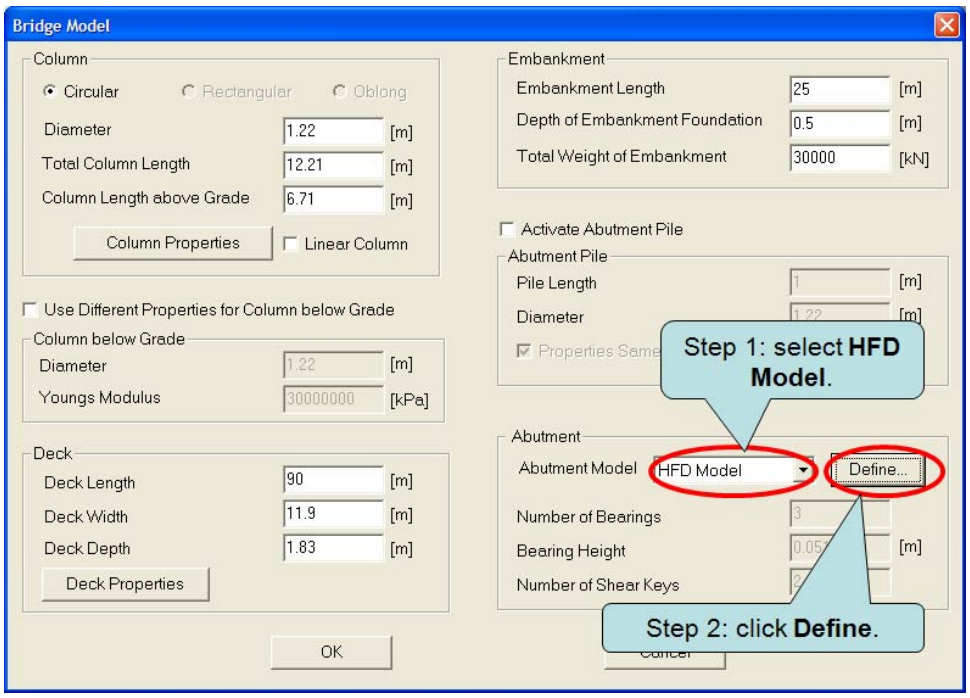
To define an HFD abutment model, please follow the steps shown in Fig. 33.

4 Soil Parameters

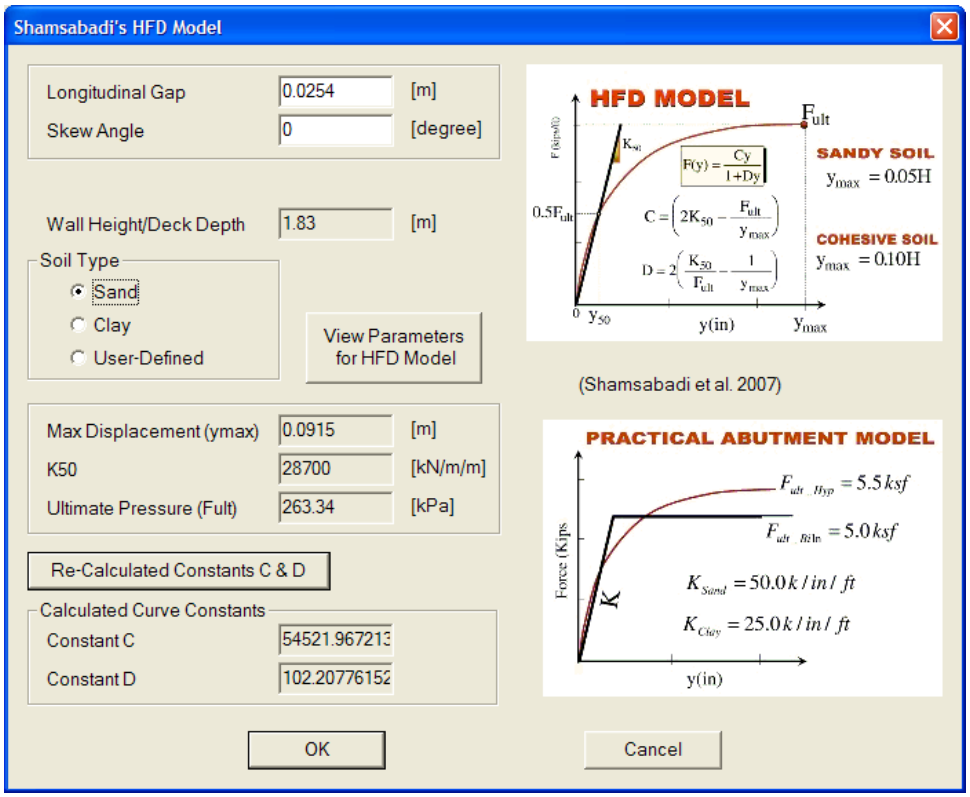
First, some important master control options are defined by clicking "**Analysis Options**" as shown in Fig. 6. This will display the interface shown in Fig. 34 below. The following modifications can be made in this window:

1. Select to keep the soil properties as defined by their linear properties, or opt to conduct nonlinear soil computations (note that the default is Linear),
2. Select from among a number of available Brick elements in OpenSees,
3. Conduct more than one earthquake simulation at a time when performing a PBEE multi earthquake record analysis,
4. Apply own weight of the soil using a global lateral stress coefficient, and a single value of Young's modulus that is user defined (this will reduce initial shear stresses in the mesh due to own weight application, but generally may have minimal impact on the subsequent earthquake computations anyway),
5. by clicking on "**Change OpenSees Options**" (Fig. 35 you can change the beam-column element type (advanced feature, please exercise with care), and
6. by clicking on "**Change Rayleigh Damping**" (Fig. 36) you can change the viscous damping characteristics of the model.

The soil parameters section (Fig. 37 below) is activated by clicking "**Soil Parameters**" in Fig. 6. . Here the horizontally stratified soil profile can be defined layer by layer (as many as 10 layers as shown in Fig. 37 below). Currently, only the cohesive soils are available to select (e.g., by clicking on the U-Clay2 section in Fig. 37, and then selecting any of the available soil types (stiff, medium and soft clay or U-Clay2 in Fig. 38).



a)



b)

Fig. 33. Steps to define a HFD abutment model: a) choosing HFD model at the bridge model window; b) HFD model window

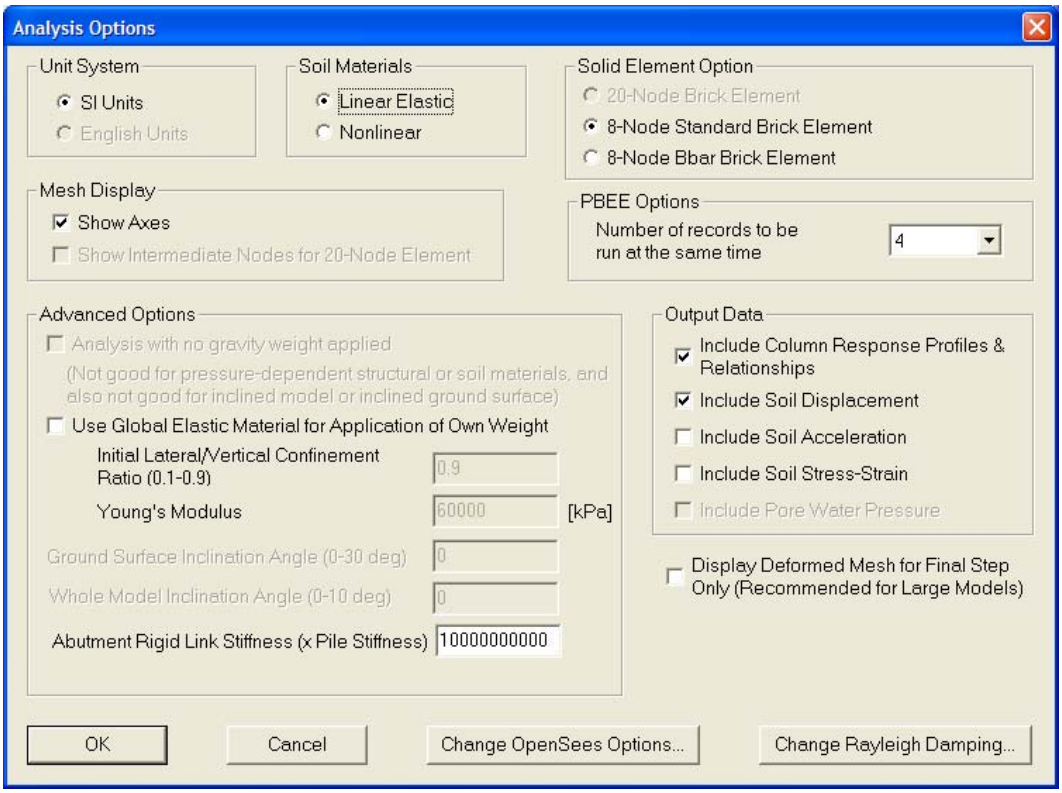


Fig. 34. Analysis options

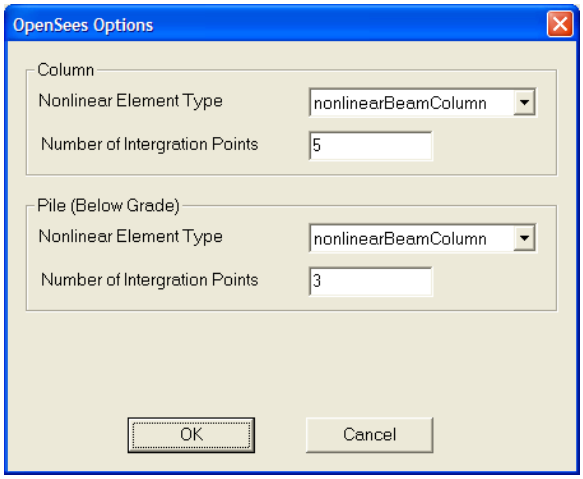


Fig. 35. Beam-column element types available for column

If you select "Activate Tension Cutoff for Cohesive Soil" as shown in Fig. 37, the soil shear strength will become negligible when volumetric stress is tensile, allowing for instance the pile to pull away without resistance from the adjacent clay on the side where tension between the pile and soil might contribute (this is an advanced feature and should be exercised with care).

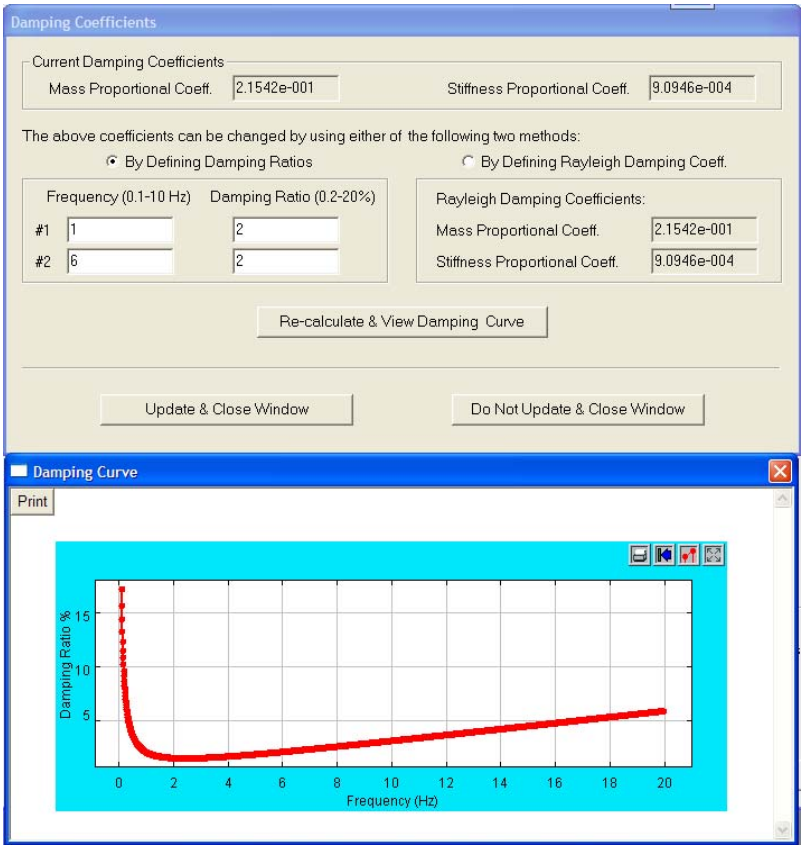


Fig. 36. Rayleigh damping coefficients

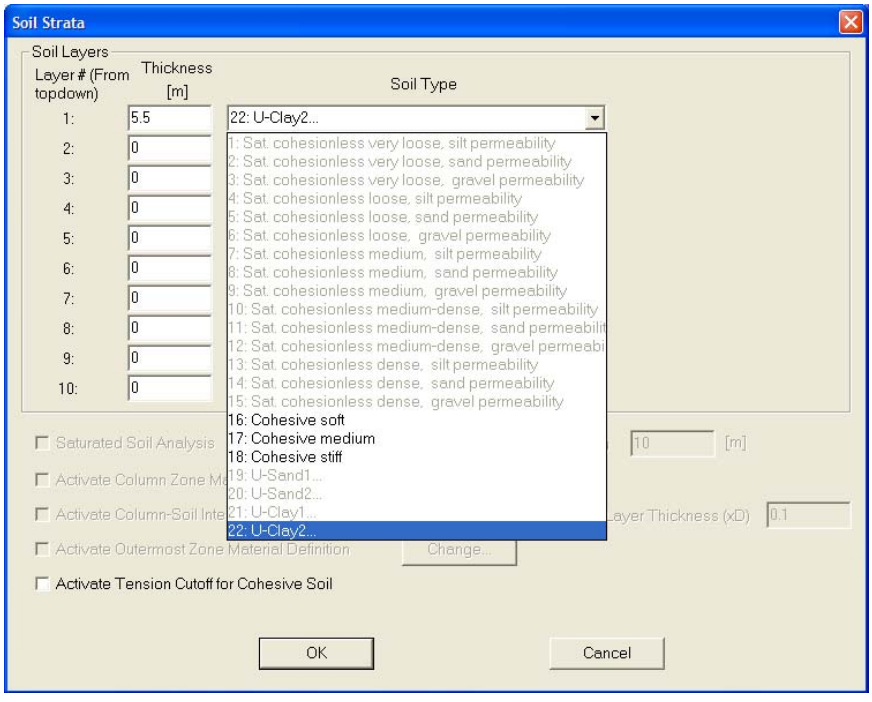


Fig. 37. Soil strata definition

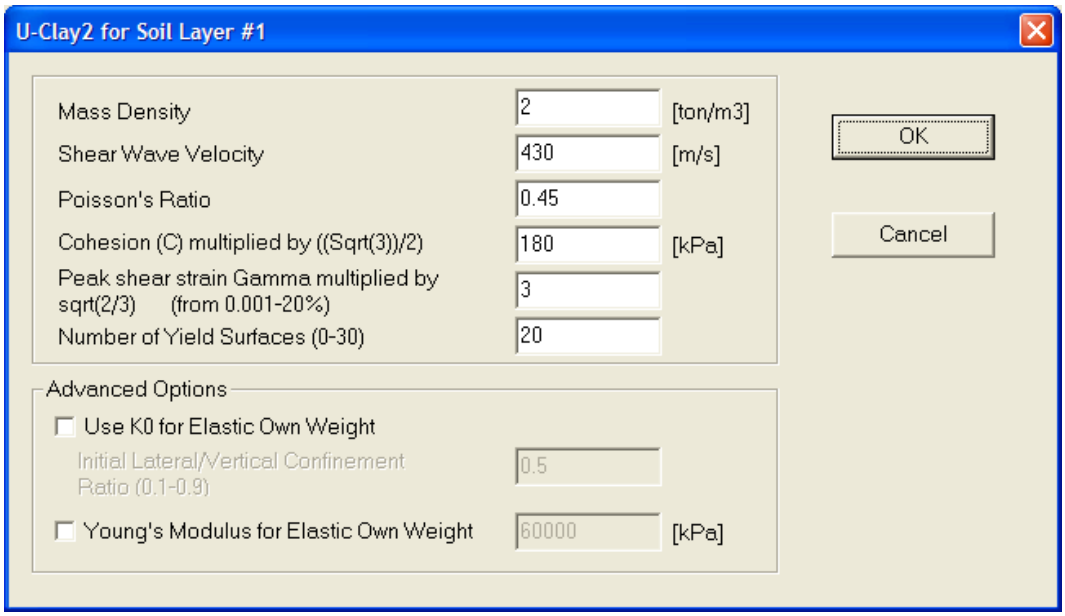


Fig. 38. User-defined clay material U-Clay2

The properties of the cohesive stiff, medium, and soft clay models are shown in Table 9 below:

Table 9. Clay material properties

	Soft Clay	Medium Clay	Stiff Clay
Mass density (ton/m^3)	1.3	1.5	1.8
Reference shear modulus (kPa)	1.3x10^4	6.0x10^4	1.5x10^5
Reference bulk modulus (kPa)	6.5x10^4 kPa	3.0x10^5 kPa	7.5x10^5
Cohesion (kPa)	18	37	75
Peak shear strain	0.1	0.1	0.1
Friction angle (degree)	0	0	0
Pressure dependent coefficient	0	0	0

The above-mentioned soil models are based on earlier research (Elgamal et al 2003; Elgamal and Lu 2009; Elgamal et al 2009; Elgamal et al 2009b; Elgamal 2010; Lu 2006; Yang et al 2003).

Finally, the soil meshing procedures are discussed in Appendix A.

5 Pushover & Eigenvalue Analyses

To conduct a pushover analysis, a load pattern must be defined (please follow the steps shown in Fig. 39). The load pattern window is shown in Fig. 40. Please see Appendix B for pushover examples.

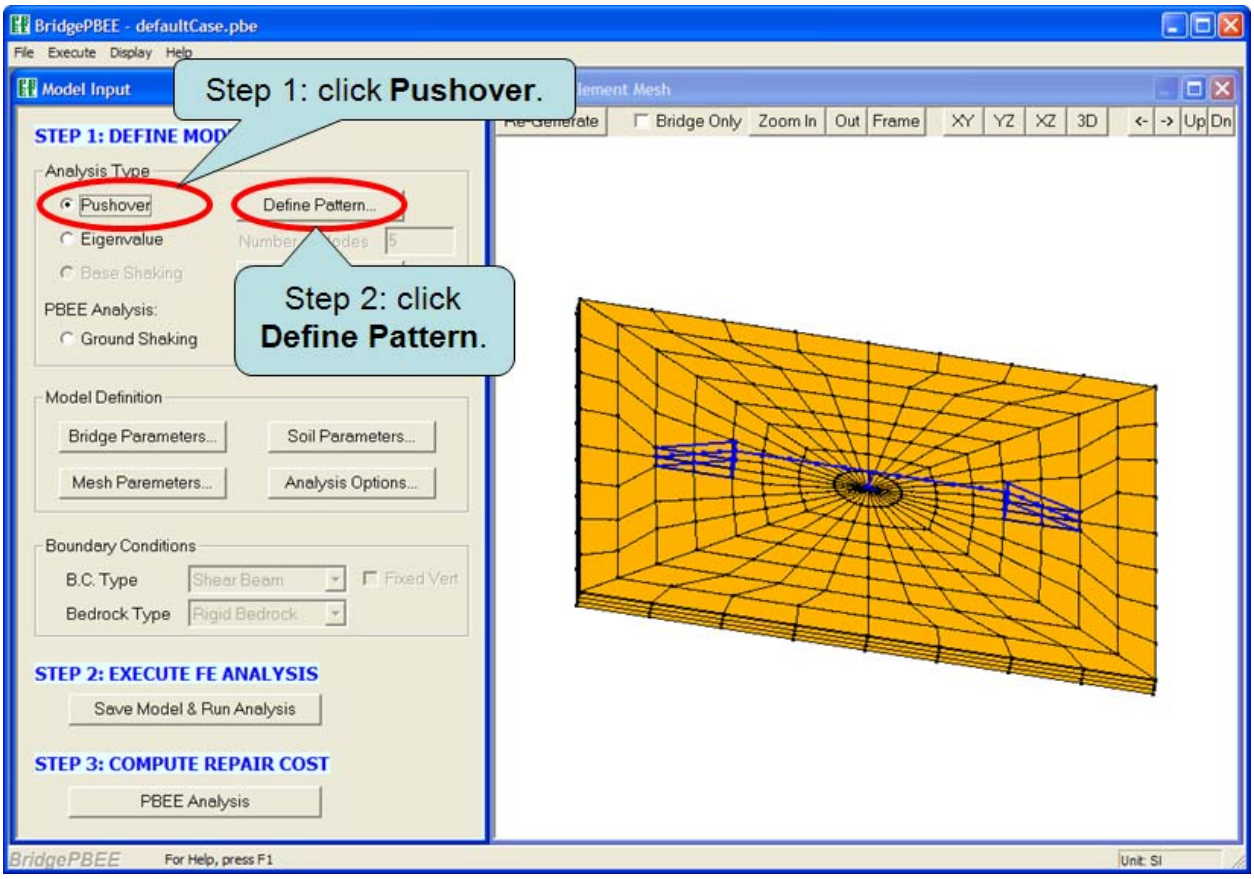


Fig. 39. Steps to define a load pattern for pushover analysis

5.1 Load Pattern

The pushover options include monotonic pushover as well as pushover by a user-defined loading pattern (U-Push). Please see the next section for how to define a U-Push file.

Two methods of pushover are available (Fig. 40): force-based and displacement-based. If **Force-Based Method** is chosen, please enter the parameters of force increment (per step): **Longitudinal (X) Force**, **Transverse (Y) Force**, **Vertical (Z) Force**, **Moment @ X**, **Moment @ Y**, and **Moment @ Z**.

If **Displacement-Based Method** is chosen, please enter the displacement increment parameters (per step): **Longitudinal Displacement**, **Transverse Displacement**, **Vertical Displacement**, **Rotation around X**, **Rotation around Y**, and **Rotation around Z**.

The pushover load/displacement linearly increases with step in a monotonic pushover mode. The load/displacement is applied at the column top.

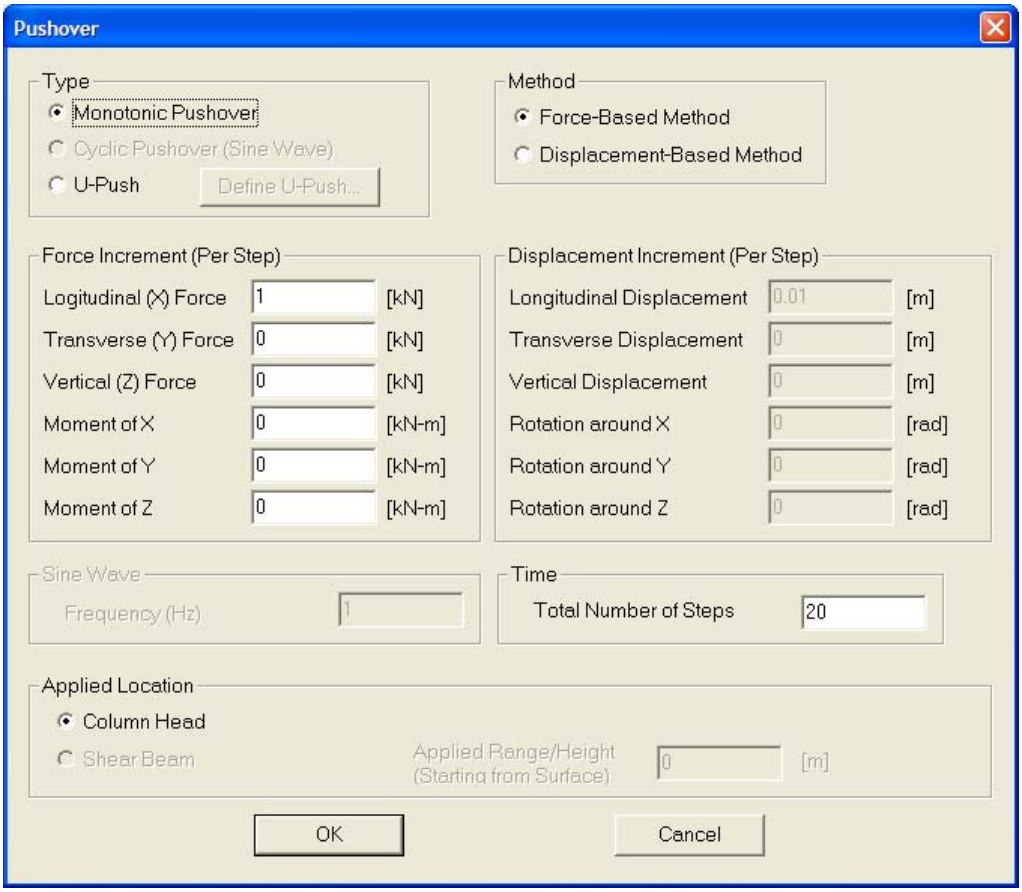


Fig. 40. Load pattern for pushover analysis

5.1.1 Pushover by User-Defined Load Pattern (U-Push)

To define your own load pattern (U-Push), please follow the steps shown in Fig. 41. The U-Push window is shown in Fig. 42. Click **Select/Change Pushover File** to change file. The user-defined pushover file should contain single-column data.

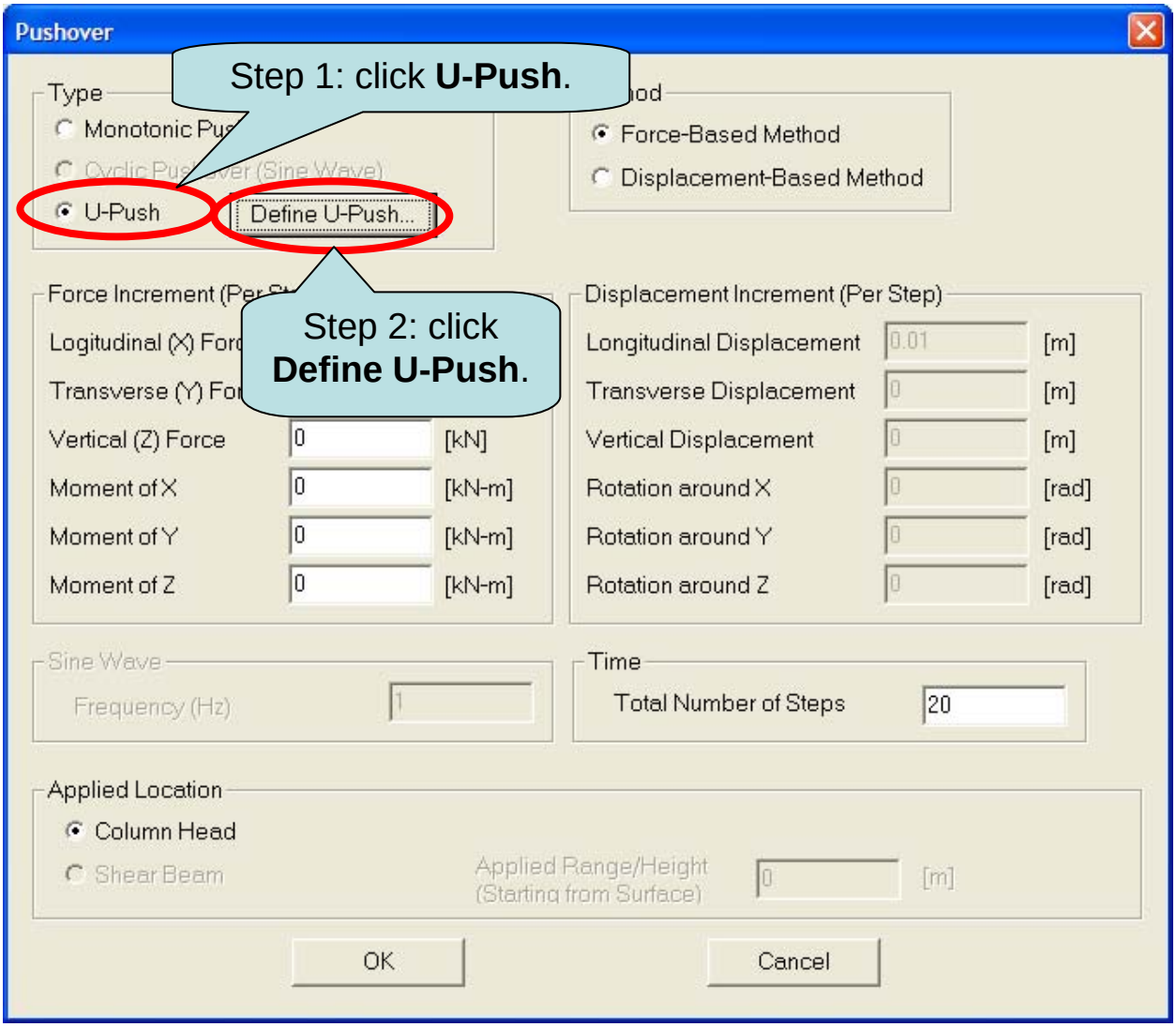


Fig. 41. Steps to define a user-defined load pattern (U-Push)

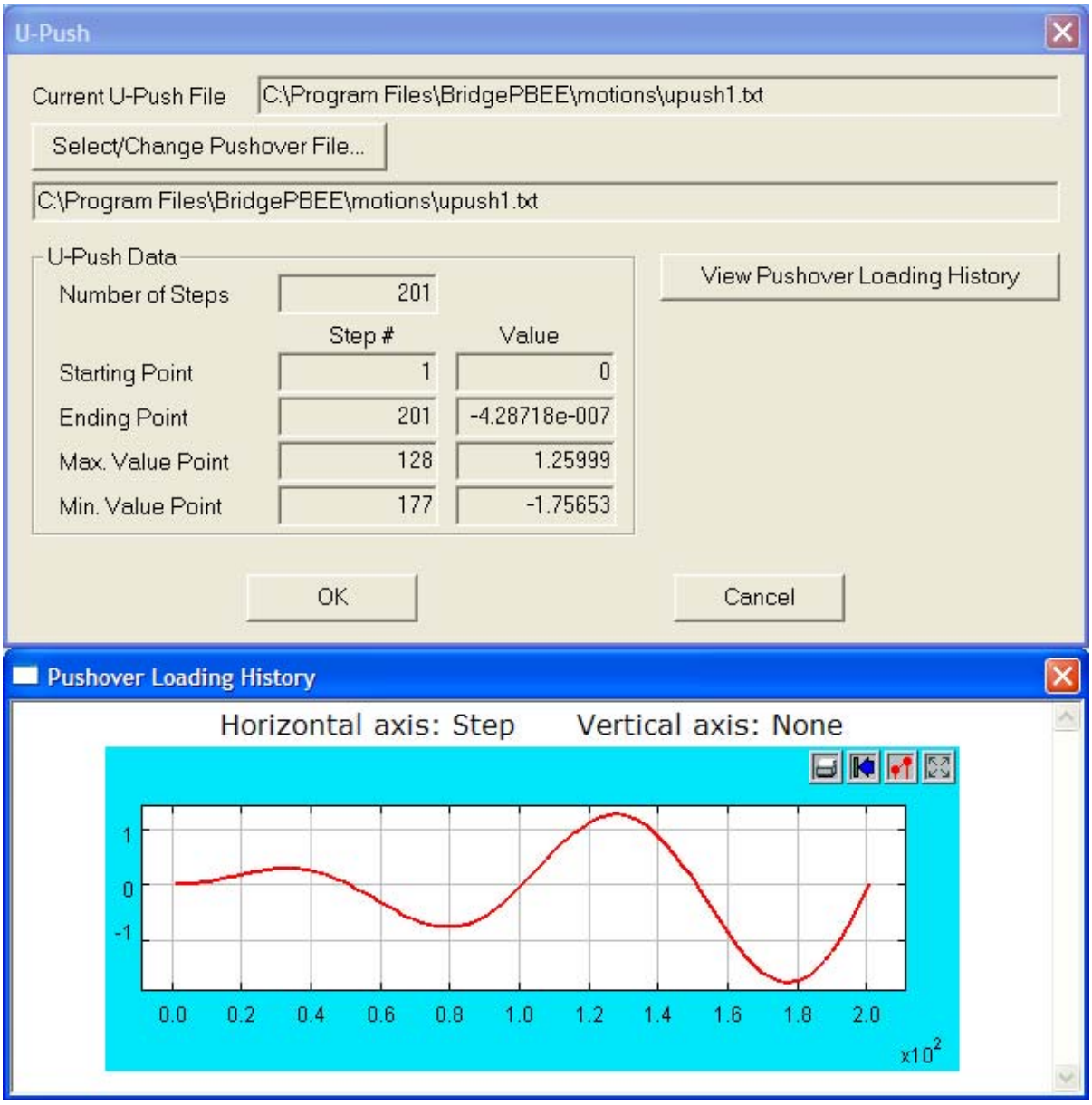


Fig. 42. Example of user-defined pushover load pattern (U-Push)

5.2 Output for Pushover Analysis

Output windows for a pushover analysis include:

- Response time histories and profiles for column (and pile shaft under grade)
- Response relationships (force-displacement as well as moment-curvature) for column (and pile shaft under grade)
- Abutment response time histories
- Deformed mesh, contour fill, and animations.

5.2.1 Column Response Time Histories and Profiles

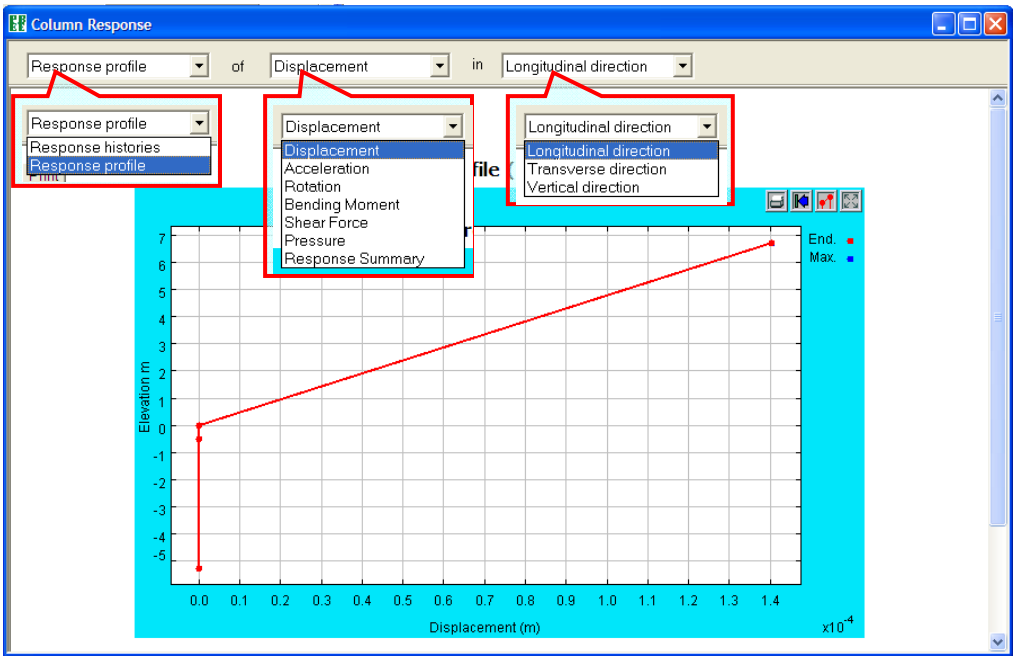


Fig. 43. Response time histories and profiles for column (and pile shaft)

5.2.2 Column Response Relationships

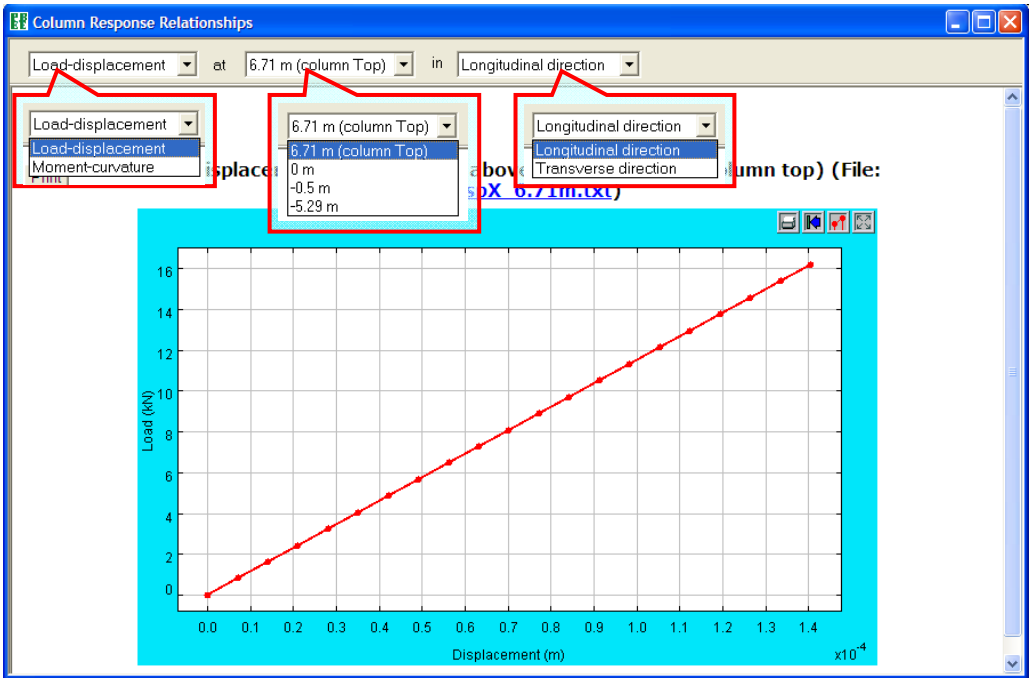


Fig. 44. Response relationships for column (and pile shaft)

5.2.3 Abutment Force-Displacement and Response Time Histories

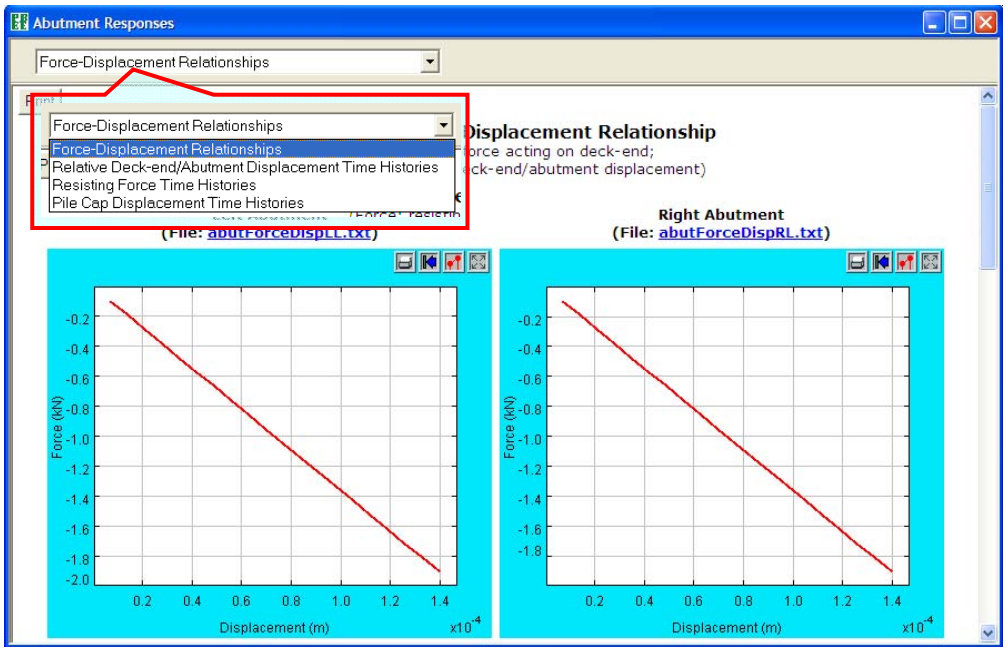


Fig. 45. Abutment response time histories

5.2.4 Deformed Mesh

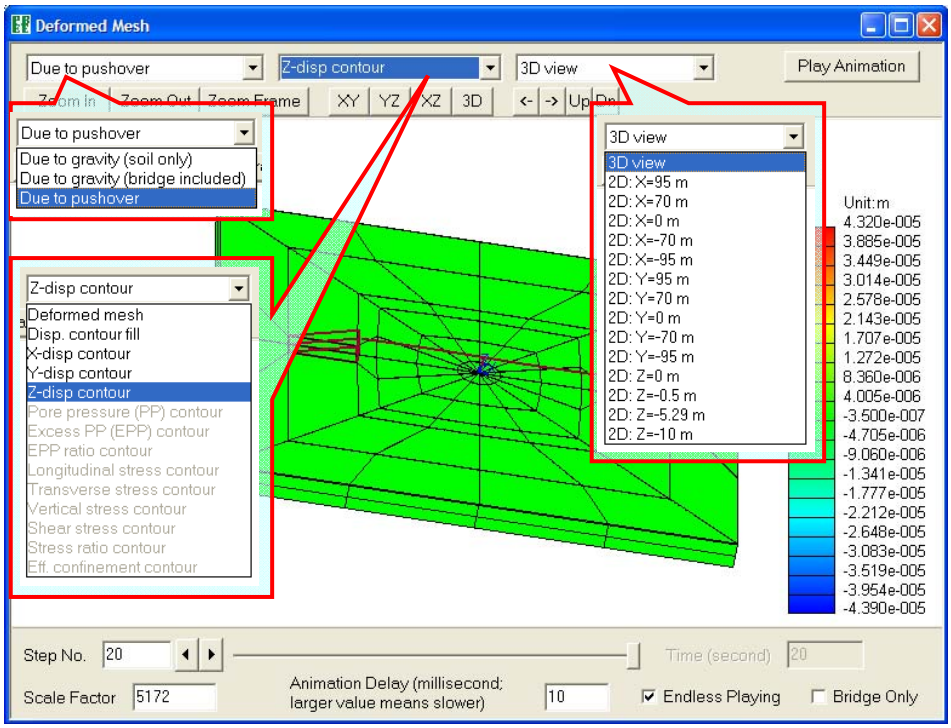


Fig. 46. Deformed mesh and contour fill

5.3 Eigenvalue Analysis

To conduct an Eigenvalue analysis, please follow the steps shown in Fig. 47 and then click **Save Model & Run Analysis**. Fig. 48 shows the output window for an Eigenvalue analysis, which can be accessed by clicking menu **Display** (Fig. 3) and then choosing **Deformed Mesh**.

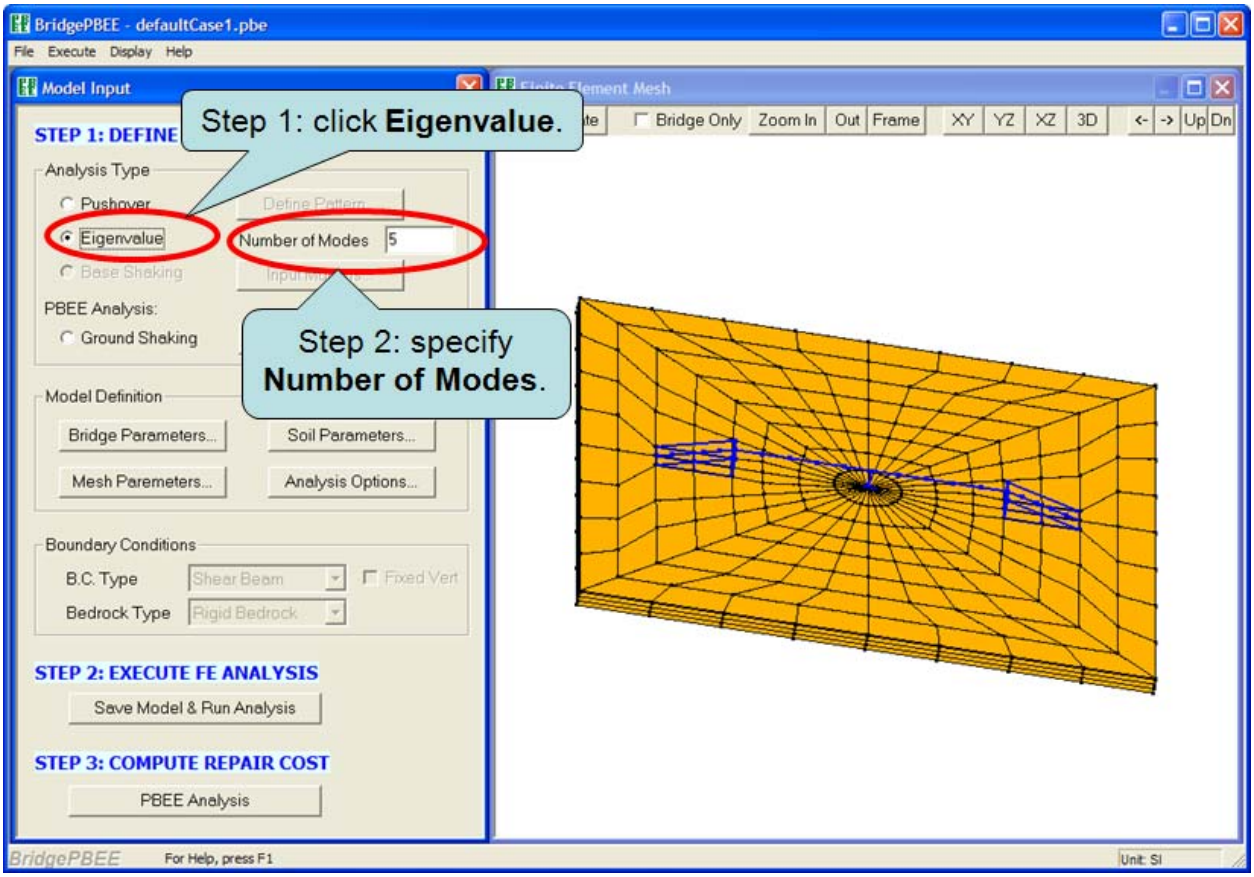
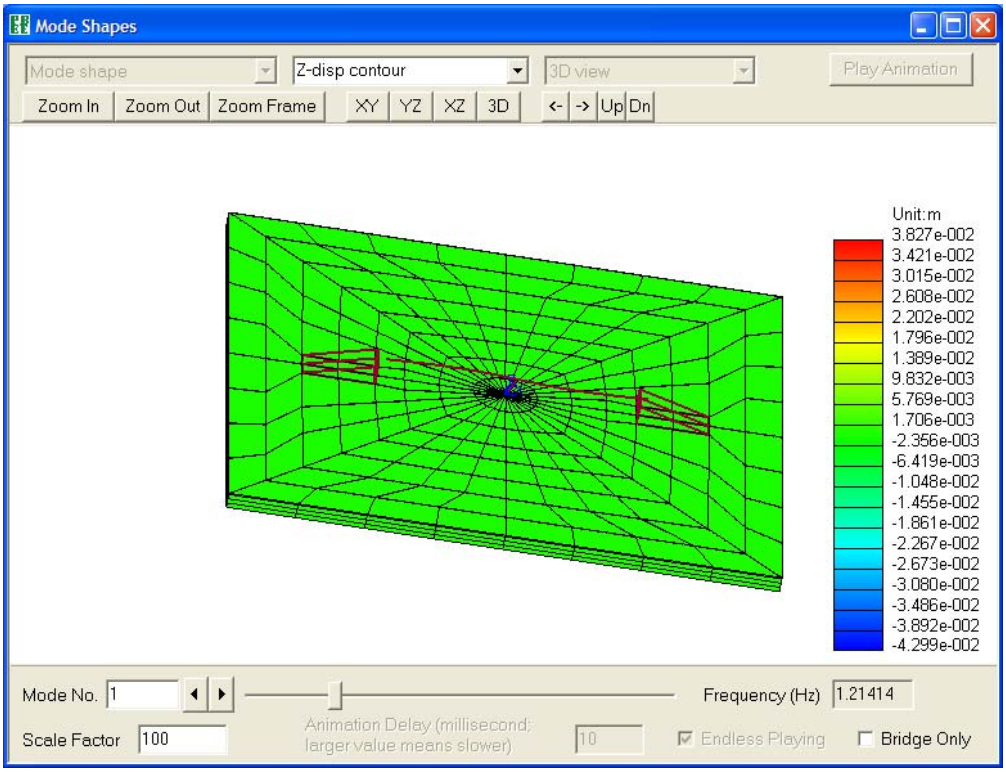
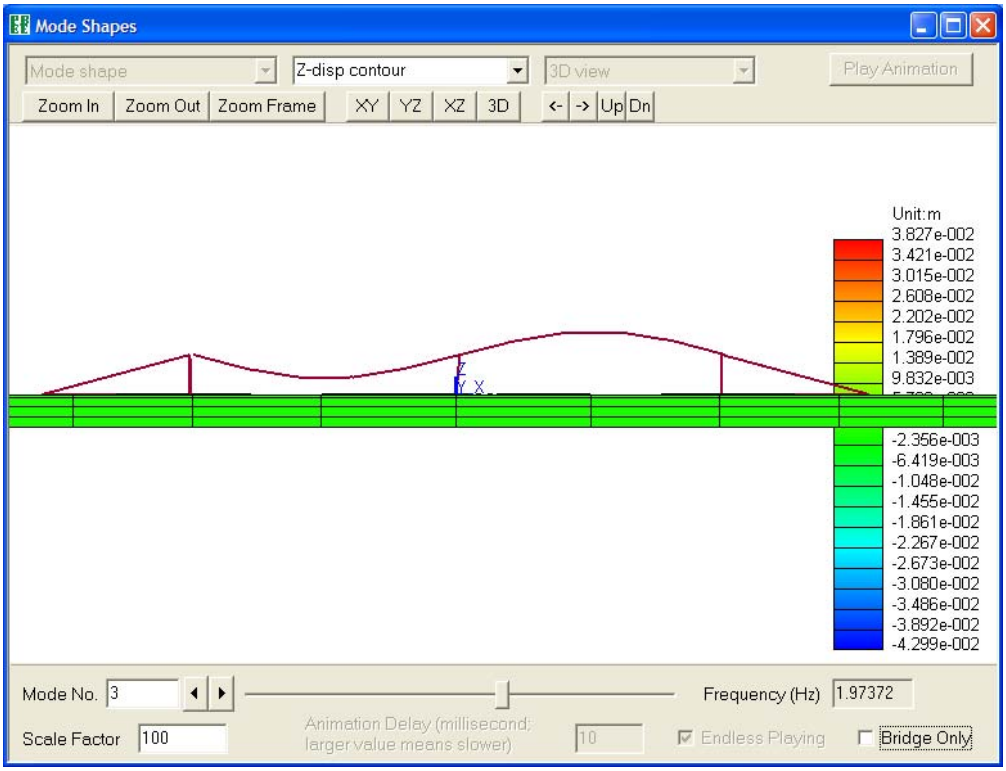


Fig. 47. Steps to perform an Eigenvalue analysis



a)



b)

Fig. 48. Sample output for an Eigenvalue analysis

6 PBEE Analysis (Ground Shaking)

To conduct a single earthquake analysis or a full PBEE analysis, the "Ground Shaking" option under Analysis Type (Fig. 2) is used. For that purpose, the input earthquake excitation(s) must be specified. Input files at http://peer.berkeley.edu/bridgepbee/ that exercise this option include Examples 2-5. If only one earthquake record is selected out of a specified ensemble (suite) of input motions, then a conventional single earthquake analysis will be performed.

6.1 Theory and Implementation of PBEE Analysis

In the user interface, an implementation of the Pacific Earthquake Engineering Research (PEER) Center's performance-based earthquake engineering framework (Cornell and Krawinkler, 2000) is used to generate probabilistic estimates of repair cost and repair time. The PEER PBEE framework utilizes the total probability theorem to compute the desired probability distributions by disaggregating the task into several intermediate probabilistic models with different sources of randomness and uncertainty. The hazard model uses earthquake ground motion data to determine an intensity measure (IM). The demand model uses response from dynamic analysis to determine an engineering demand parameter (EDP). The damage model connects the EDP to a damage measure (DM). Then, the DM is linked to consequences that are termed the decision variables (DV). Repair cost and repair time can be thought of as two possible decision variable (DV) outcomes characterized probabilistically by the framework.

The complete analysis is accomplished using the local linearization repair cost and time methodology (LLRCAT), described by Mackie et al. (2010) and depicted conceptually in Fig. 49. In the LLRCAT methodology, each bridge system is disaggregated into independent structural or non-structural components or subassemblies defined as performance groups (PGs) that are damaged, assessed, and repaired together using a specific combination of different repair methods. Demands on the bridge system (and components) are determined using 3D nonlinear time history analysis under multiple-component earthquake excitation. The damage in each of the PGs is characterized according to several discrete damage states (DSs) that are defined by distributions of critical EDPs.

A feature of the LLRCAT implementation used is the introduction of a repair model between the original PEER abstraction of DM and DV. Jumping directly from DMs to repair costs is difficult to accomplish, because it skips over the details of repair design and the variability of cost and time estimating. Creating these two additional models makes it easier to implement a step-by-step procedure for defining the models. The repair model and cost model are created through the process of schematic design of repairs and estimating the costs of those designs. Different repair methods are employed for the various damage states of each PG or bridge component. The repair methods for each PG require a combination of several repair quantities (Qs). Repair quantities for all PGs are then combined with due consideration of the correlation between components. Repair costs (RC) are obtained through a unit cost (UC). Repair times (RT) are obtained through a production rate (PR). The PRs are in terms of crew working days (CWD), representing one working day for a normal sized crew and can be combined later by construction management experts to obtain total site construction times.

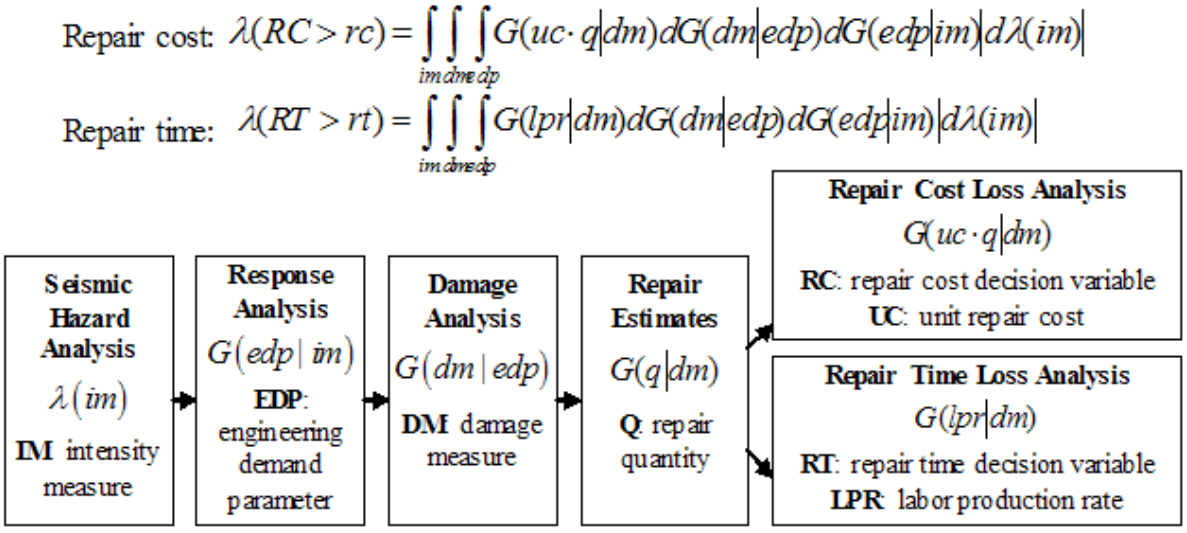


Fig. 49. Schematic procedure of the LLRCAT methodology for a single bridge component

The characterization and visualization of the ground motion suites using different choices of IMs will be discussed in Section 6.3. The FEM, parameter selection, analysis options, and outcomes that generated EDPs were similarly covered in Section 3. The bridge is then broken down into performance groups (PGs) for each major bridge superstructure, substructure, and foundation component. Each performance group represents a collection of structural components that act as a global-level indicator of structural performance and that contribute significantly to repair-level decisions. Performance groups are not necessarily the same as load-resisting structural components. For example, non-structural components may also form a performance group, since they also suffer damage and contribute to repair costs. The notion of a performance group also allows grouping several components together for related repair work. For example, it is difficult to separate all of the individual structural components that comprise a seat-type abutment (shear key, back wall, bearings, approach slab, etc.) as they all interact during seismic excitation and their associated repair methods are coupled. Therefore, the abutment repair group incorporates the fact that repairs to the back wall require excavation of the approach slab.

Performance groups also address the issue of potentially double counting related repair items. Some repair items require the same preparation work such as soil excavation. For example, both back wall repair and enlargement of an abutment foundation require at least 4 ft of excavation behind the back wall. If these repair items were in different PGs, then double counting the excavation would be a problem. Bundling these related repair methods within a PG allows for independent consideration of each PG. The correlation between repair items from the PGs is handled at the demand model level in the methodology. A total of 11 PGs are considered: PG1: Max column drift ratio; PG2: Residual column drift ratio; PG3: Max relative deck-end/abutment displacement (left); PG4: Max relative deck-end/abutment displacement (right); PG5: Max bridge-abutment bearing displacement (left); PG6: Max bridge-abutment bearing displacement (right); PG7: Approach residual vertical displacement (left); PG8: Approach residual vertical displacement (right); PG9: Abutment residual pile cap displacement (left); PG10: Abutment

residual pile top displacement (right); PG11: Column residual pile displacement at ground surface.

Discrete DSs are defined for each PG. Damage states are numbered sequentially in order of increasing severity. The DS0 damage state corresponds to the onset of damage when repair costs begin to accumulate. An upper limit to the quantities and costs is called DS∞, because it corresponds to the most severe possible damage state for the elements in a PG. DS∞ usually corresponds to complete failure and replacement of all the elements in the entire PG. The DSs are connected to structural demands obtained from finite element analysis results by way of an EDP specific to each PG. The repair quantities associated with each DS are developed more fully in the definition of the damage scenarios. All the PGs and DSs are linked to a single EDP in this implementation.

Based on previous work, the methodology was calibrated for defining post-earthquake performance of select bridges that fall within the class of ordinary post-tensioned, box girder, reinforced concrete highway overpasses (Mackie et al. 2011). The three major components required for this calibration were damage scenarios that describe particular instances of earthquake damage, schematic design of bridge repairs to address the state of damage in the scenarios, and the link between repair design, methods and procedures, and subsequent quantities. There is a direct link between damage scenarios and the repair, i.e., there is a single repair procedure for a single state of damage. The repair quantity results were parameterized in terms of basic bridge geometry and properties so that they can be used to extrapolate loss modeling for other bridges in the same class (such as those that can be built within the user interface).

Data for time and monetary repair costs were obtained by estimating the costs of the damage and loss scenarios using published Caltrans construction estimation data, case studies from previous earthquakes, and interviews with Caltrans bridge engineers. Monetary costs were adjusted to 2007 values based on Caltrans cost index data. Repair costs are estimated for each damage scenario based on quantities of each repair item. Cost estimates accounted for variations in unit cost, and the details involved in estimating a combination of repairs together. The benefit of separating the Qs from costs is that the unit cost model is easily updated for new years of data, local economic conditions, site accessibility, and incentives.

Normalized costs of repair are obtained by using the repair cost ratio (RCR) between the cost of repair and the cost of replacement cost (does not include demolition). It is shown in %, and it can range between 0 and some number higher than 100% (there is no reason why it is bounded by replacement cost; this is purely an owner/operator decision). This ratio is useful for comparing the performance of different bridge design options for new construction. For the evaluation of existing structures, the RCR including demolition costs might be more useful. Constructing a new bridge on the same site after an earthquake would require both demolition of the damaged bridge and construction of its replacement. The costs of new construction used in the interface come from Caltrans bridge cost estimates used for planning purposes. They are based on the deck and type of construction, providing a range of cost/SF of deck area, circa 2007 to be consistent with the repair data used.

Repair time for the bridge can be expressed either as an approximation of repair duration or repair effort. The repair effort represents the total number of crew-workdays (CWD) required to complete the task. This is different from repair duration, which counts the total duration of the repair project. The repair duration includes the effect of scheduling concurrent on-site construction processes, while the repair effort does not. The repair duration can vary based on the amount and type of concurrent construction processes, schedule dependencies, availability of labor, and whether or not contract incentives are provided in order to decrease duration. Repair times are also computed on the basis of each repair quantity Q. For any repair item, a probability of 50% that Q > 0 indicates that the associated repair time should be added to the total repair time for the project.

6.2 Input Necessary for User-defined PBEE Quantities

If the user is interested in providing user- or project-specific information in a PBEE analysis, the following paragraphs describe the data needed by the interface to execute the PBEE analysis and post-process the results. Performance groups need to be defined for each important component or subassembly of the system that has potential repair consequences. Performance groups are defined in terms of a single EDP that characterizes the response of this PG. Once this EDP metric has been defined and time history analysis performed to obtain a distribution of EDP realizations for different ground motions, the PBEE methodology can be implemented. The PBEE methodology requires definition (by the user) of discrete damage states for each PG, a repair method with associated repair quantities for each discrete DS for each PG, and the corresponding costs and times required to execute the repair method.

The damage states are discrete and supplied in the form of what is commonly called a fragility curve. This is a misnomer however, because the information required is the value of the EDP (not IM) required to trigger different probabilities of exceeding the given discrete DS. It is often assumed that said curves are well described by the lognormal probability distribution and therefore, the only parameters required are the two lognormal distribution parameters: lambda and beta. Lambda is the median and beta is the lognormal standard deviation. A PG can have as many discrete DS as are required to cover the full range of possible responses experienced by the PG. These should be input as is shown in Section 6.5.1 below.

Once the different states of damage have been established, damage scenarios need to be generated that show different possible “snapshots” of damage that the structure may be in after an earthquake event. Once these scenarios have been generated (note the scenarios need to be detailed and include exact descriptions of the extent and depth of damage), they can be used to decide what repair method would be appropriate for each PG or group of PGs. Such information is specific to the type of structure, the discrete DSs, and the PGs. It is really only obtainable from experts with past experience designing repair procedures given a damage scenario or snapshot. Once the repair methods have determined, specific details about the repair quantities (specific meaning square footage of deck, cubic yards of concrete, etc.) can be specified. The current data employed in the interface has repair quantities parameterized in terms of the common bridge design and geometric parameters, making it possible to solve for a variety of bridges within class.

However, any changes beyond these configurations would require numerical values for all the repair quantities to be input.

It is assumed that the repair quantity estimates for each PG and DS are also random quantities and can be described by a mean (or median) value and a coefficient of variation or lognormal standard deviation. In the interface, beta has been set as 0.4, but could be modified by the user in the future (if so desired). The repair quantities may then be handed over to a cost estimator, who would have the ability to access historical pricing and bid information. In addition, the type and magnitude of each repair quantity would correspond to standard DOT estimates and specifications procedures. Each repair quantity can then be bid or an estimation of cost and effort/time/production rate made. These unit costs and production rates are also random quantities and can be described by a mean (or median) value and a coefficient of variation or lognormal standard deviation. The values currently in the interface all have a beta of 0.2, but could again be set by the user if desired. See more details about PERT criteria for the production rates in Mackie et al. (2008).

Modifying the default PBEE quantities (repair quantities, unit costs, and production rates) is detailed in Appendix E.

6.3 Definition/specification of PBEE input motion ensemble (suite)

6.3.1 Available Ground Motions

Four ground motion data sets are available:

Motion Set 1: This PBEE motion ensemble were obtained from the PEER NGA database (http://peer.berkeley.edu/nga/) and consist of 100 3D input ground motions triplets, sorted into 5 bins. Each motion is composed of 3 perpendicular acceleration time history components (2 lateral and one vertical). These motions were selected through earlier efforts (Gupta and Krawinkler, 2000; Mackie et al., 2007) to be representative of seismicity in typical regions of California. The motions are divided into 5 bins of 20 motions each with characteristics: i) moment magnitude (Mw) 6.5-7.2 and closest distance (R) 15-30 km, ii) Mw 6.5-7.2 and R 30-60 km, iii) Mw 5.8-6.5 and R 15-30 km, iv) Mw 5.8-6.5 and R 30-60 km, and v) Mw 5.8-7.2 and R 0-15 km. The engineering characteristics (IMs and time histories) of each motion and of the ensemble overall may be viewed directly within BridgePBEE.

Motion Set 2: These motions (160 in total) are developed by Dr. Mackie from the 80 motions of Motion Set 1 (excluding the 20 motions in bin v) above), to account for the more accurate site classifications (NEHRP C and NEHRP D) in NGA. The magnitude, distance, and spectral shape were intended to be similar to the previous bins (and indeed all of the previous motions appear in either the NEHRP C or NEHRP D bins now). As such, these motions are divided into 8 bins (compared to the 4 bins of Motion Set 1).

Motion Set 3: These motions (80 in total) are labeled Broadband_* (separated into the two bins, Broadband rock and Broadband soil) as developed by Dr. Jack Baker for PEER. Additional information about these motions is available at the website: http://peer.berkeley.edu/transportation/projects/ground-motion-studies-for-transportation-systems/

Motion Set 4: These motions (260 in total) include the above Motion Set 2 and Motion Set 3 as well as the additional bin v.) (near fault motions) of Motion Set 1.

All of the above 4 ground motion data sets were resampled to a sampling frequency of 50 Hz (regardless of whether initial sampling frequency was 100 or 200 Hz) due to the computational demands of running full ground-structure analyses for an ensemble of motions. Standard interpolation methods were used to resample the time domain signals (so that the signal shape is preserved). The resampled records were then baselined to remove any permanent velocity and displacement offsets. Baselining was accomplished using a third order polynomial fitted to the displacement record.

6.3.2 Specifications of PBEE Input Motions

To conduct a PBEE analysis, input motions must be defined (please follow the steps shown in Fig. 50). The window to define PBEE input motions is shown in Fig. 51. To unselect all motions, click **De-select All.** To see all motions, click **Select All** (same button as **De-select All**). The dropdown list of **Randomly Choose Records for Each Bin** will randomly select a certain number of input motions from each bin (the input motions are categorized by bin).

Double-click any record to view its intensity measures and response spectra (Fig. 52). SRSS stands for Square Root of Sum of Squares of the 2 horizontal components. Click **Display Intensity Measures** (Fig. 51) to view the histogram and cumulative distribution plots for whole input motion set (Fig. 53). The intensity measures include:

- PGA (Peak Ground Acceleration)
- PGV (Peak Ground Velocity)
- PGD (Peak Ground Displacement)
- $D_{5\text{-}95}$ (Strong Motion Duration)
- CAV (Cumulative Absolute Velocity)
- Arias Intensity
- SA (Spectral Acceleration; assuming 1 second period)
- SV (Spectral Velocity), SD (Spectral Displacement)
- PSA (Pseudo-spectral Acceleration)
- PSV (Pseudo-spectral Velocity)

The strong motion duration ($D_{5\text{-}95}$) is defined according to the time domain bounded by the 5% and 95% cumulative Arias intensity of the record. All of the spectral intensity measures are defined at an effective viscous damping of 5% unless otherwise noted.

In the histogram window (Fig. 53), click **Display Intensity Measures Values** to view the intensity measures listed in text format (Fig. 54). The user can copy and paste to her/his favorite text editor such as MS Excel (in Fig. 54, right-click and then click **Select All** to highlight, and then right-click and then click **Copy** to copy to the clipboard).

To incorporate user-defined input motions, please see Appendix C.

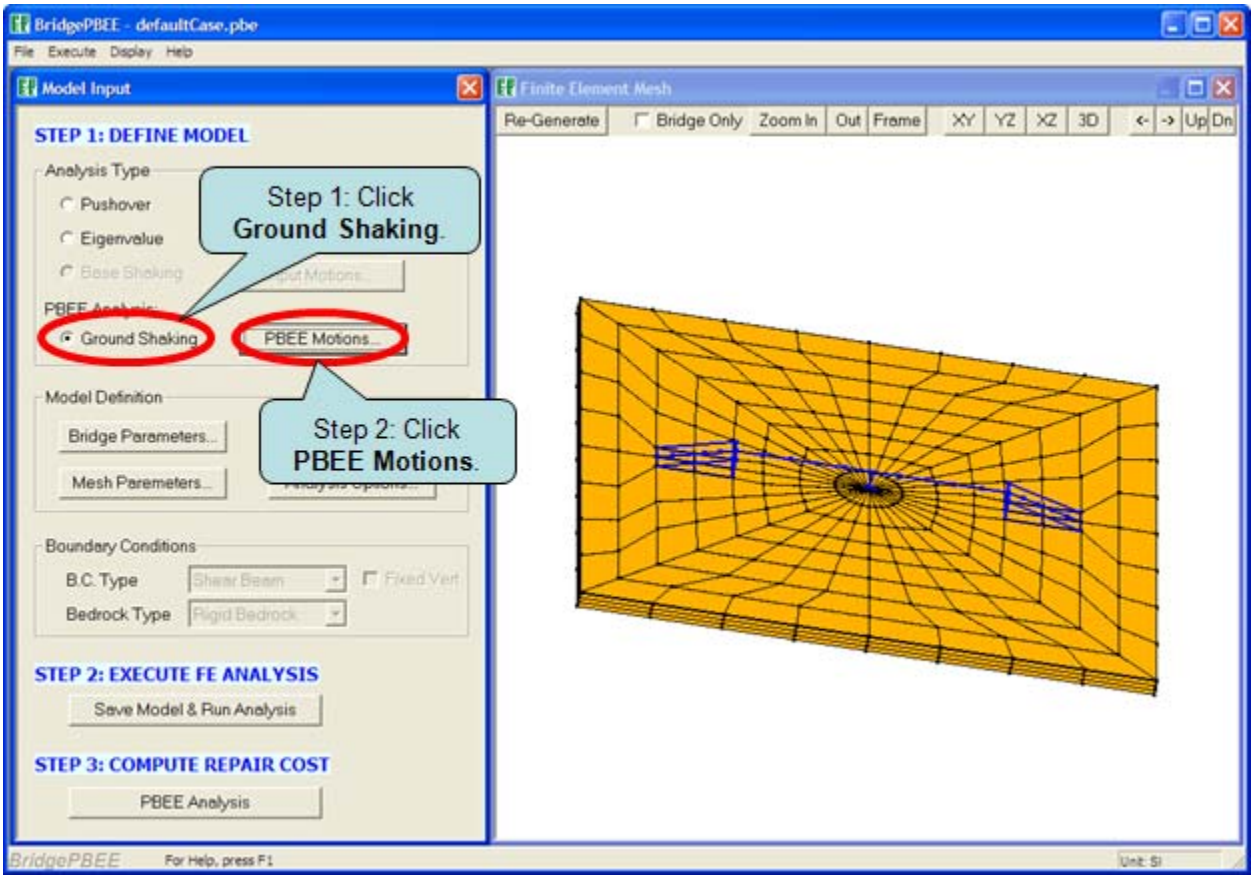


Fig. 50. Steps to define PBEE motions

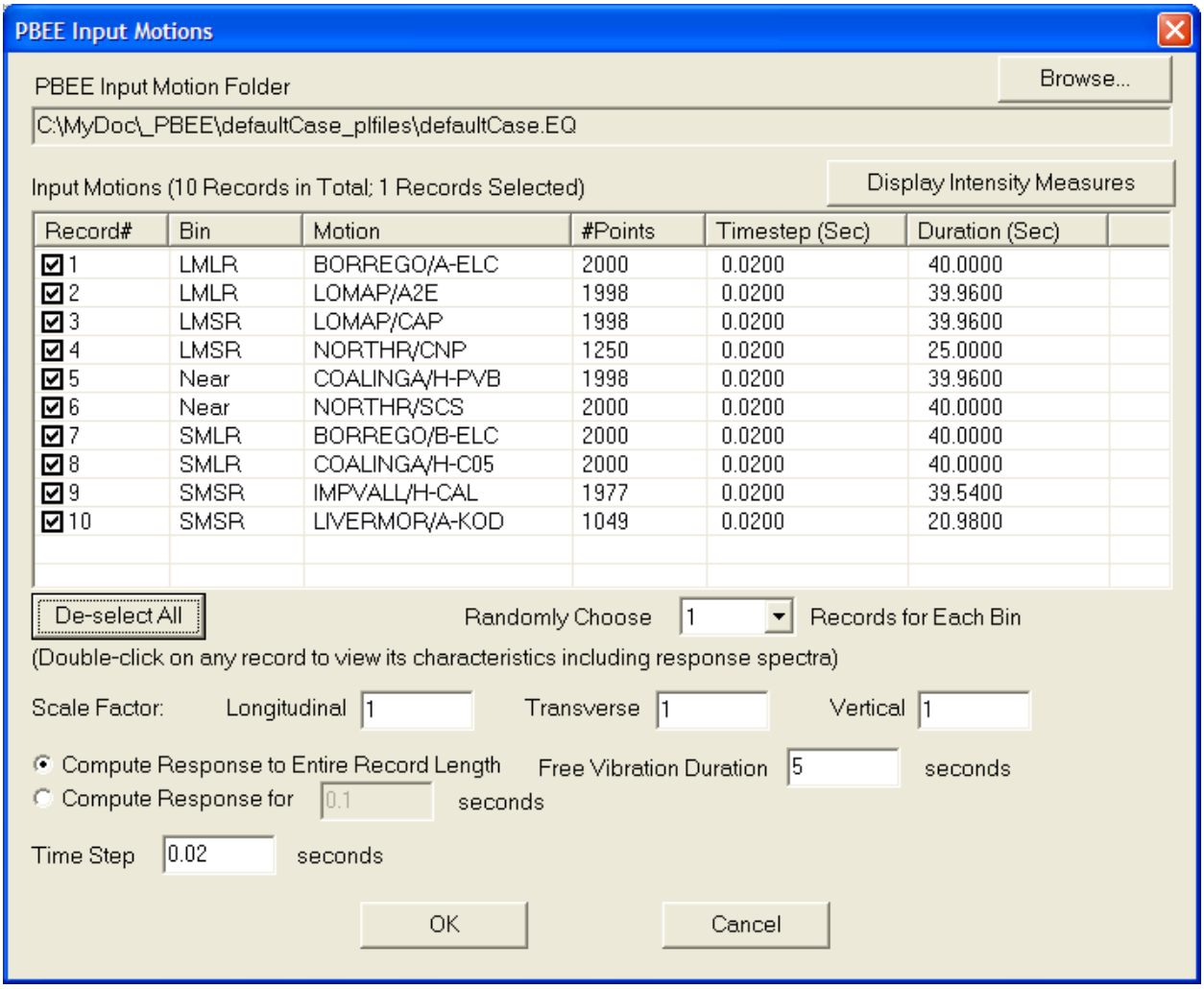
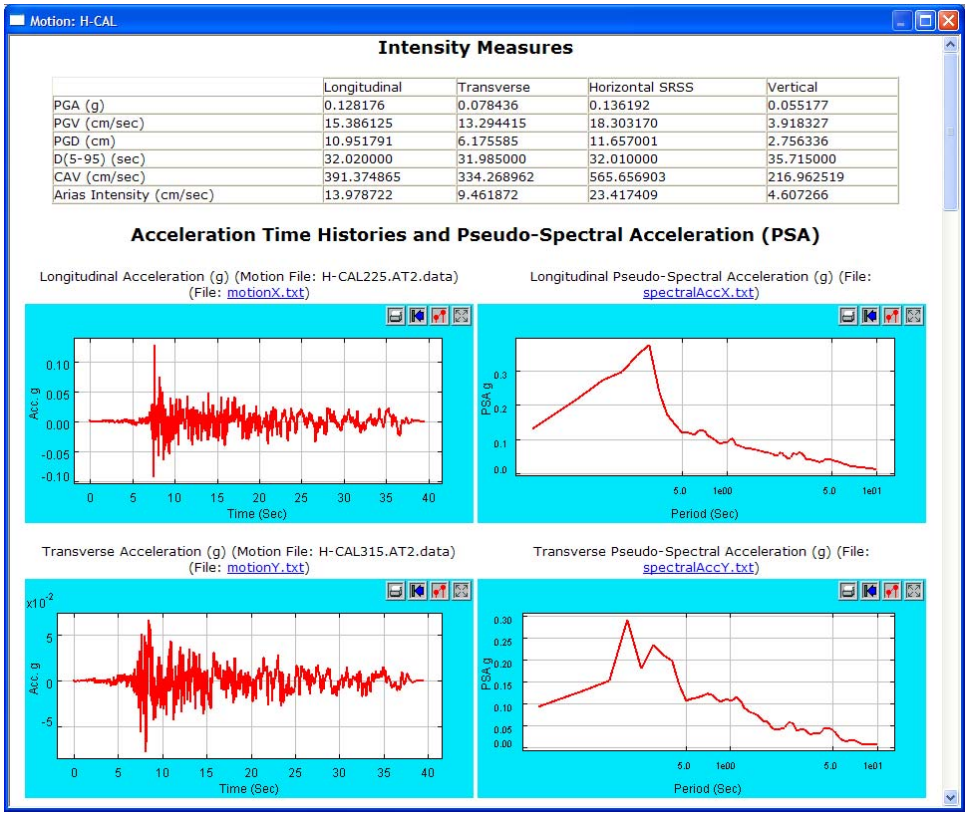

PBEE Input Motions
PBEE Input Motion Folder
Browse...
C:\MyDoc_PBEE\defaultCase_plfiles\defaultCase.EQ
Input Motions (10 Records in Total; 1 Records Selected)
Display Intensity Measures
Record# Bin Motion #Points Timestep (Sec) Duration (Sec)
1 LMLR BORREGO/A-ELC 2000 0.0200 40.0000
2 LMLR LOMAP/A2E 1998 0.0200 39.9600
3 LMSR LOMAP/CAP 1998 0.0200 39.9600
4 LMSR NORTHR/CNP 1250 0.0200 25.0000
5 Near COALINGA/H-PVB 1998 0.0200 39.9600
6 Near NORTHR/SCS 2000 0.0200 40.0000
7 SMLR BORREGO/B-ELC 2000 0.0200 40.0000
8 SMLR COALINGA/H-C05 2000 0.0200 40.0000
9 SMSR IMPVALL/H-CAL 1977 0.0200 39.5400
10 SMSR LIVERMOR/A-KOD 1049 0.0200 20.9800
De-select All
Randomly Choose 1 Records for Each Bin
(Double-click on any record to view its characteristics including response spectra)
Scale Factor: Longitudinal 1 Transverse 1 Vertical 1
Compute Response to Entire Record Length
Free Vibration Duration 5 seconds
Compute Response for 0.1 seconds
Time Step 0.02 seconds
OK
Cancel

Fig. 51. PBEE input motions widow



	Longitudinal	Transverse	Horizontal SRSS	Vertical
PGA (g)	0.128176	0.078436	0.136192	0.055177
PGV (cm/sec)	15.386125	13.294415	18.303170	3.918327
PGD (cm)	10.951791	6.175585	11.657001	2.756336
D(5-95) (sec)	32.020000	31.985000	32.010000	35.715000
CAV (cm/sec)	391.374865	334.268962	565.656903	216.962519
Arias Intensity (cm/sec)	13.978722	9.461872	23.417409	4.607266

Fig. 52. Intensity measures, time histories and response spectra of individual record

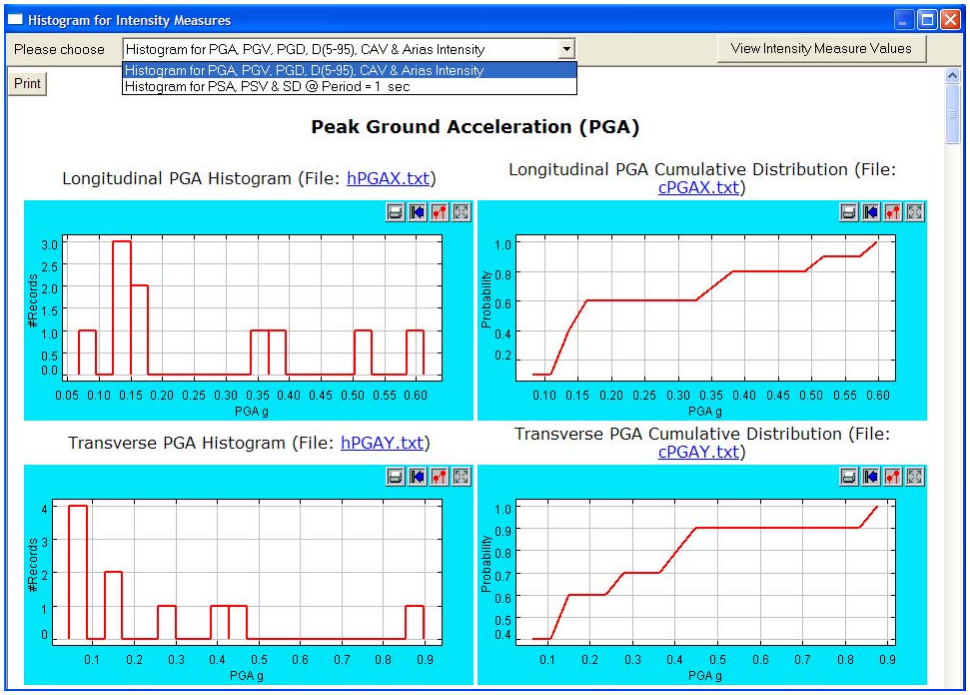

Histogram for Intensity Measures
Please choose
Histogram for PGA, PGV, PGD, D(5-95), CAV & Arias Intensity
Histogram for PGA, PGV, PGD, D(5-95), CAV & Arias Intensity
Histogram for PSA, PSV & SD @ Period = 1 sec
View Intensity Measure Values
Print
Peak Ground Acceleration (PGA)
Longitudinal PGA Histogram (File: hPGAX.txt)
Longitudinal PGA Cumulative Distribution (File:
cPGAX.txt)
Transverse PGA Histogram (File: hPGAY.txt)
Transverse PGA Cumulative Distribution (File:
cPGAY.txt)
#Records
Probability
PGA g

Fig. 53. Histogram and cumulative distribution for the whole input motion set

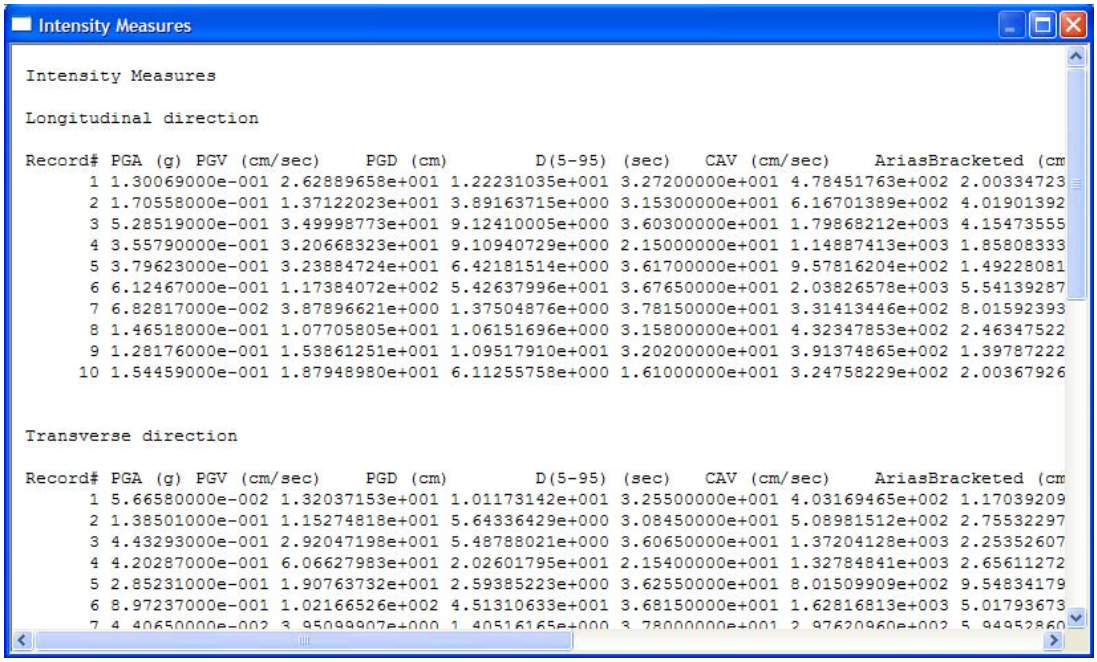


Fig. 54. Intensity Measures (IM) table for the whole input motion set

6.4 Save Model and Run Analysis

After defining the finite element model, click on "**Save model and run analysis**". The finite element computations will start, for several earthquakes at a time as specified in the "Analysis Options" (Fig. 55) window below. You can select as many as 8 records to be run at the same time in order to reduce the overall run time (for dual core machines or better). Fig. 56 shows the analysis progress for each record.

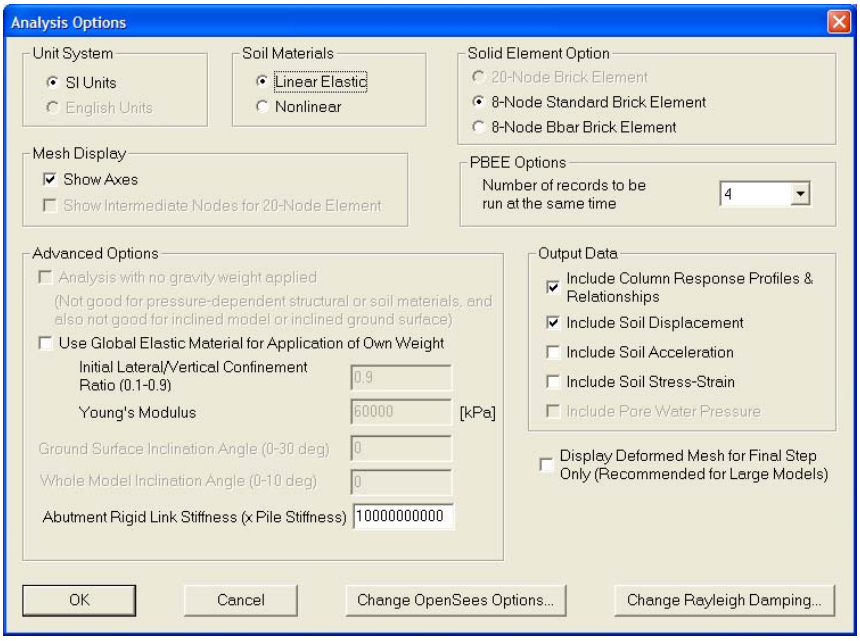


Fig. 55. Options to change number of records to be run at the same time

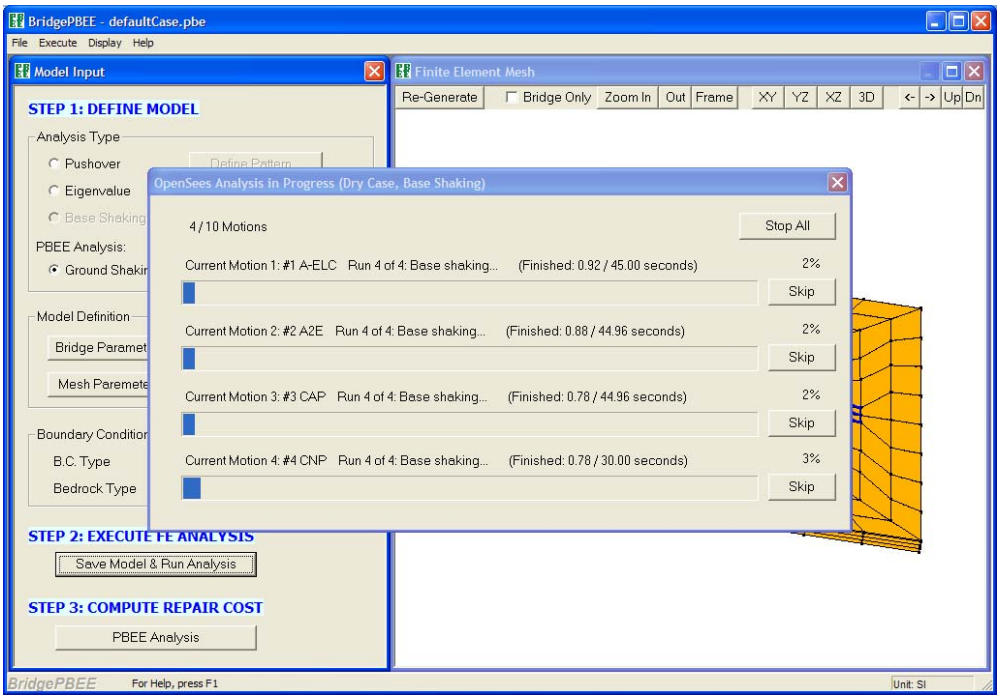


Fig. 56. OpenSees analysis in progress

6.5 PBEE Analysis

Once the FE analysis of all motions in the ensemble is complete, click **PBEE Analysis** (Fig. 2) to display:

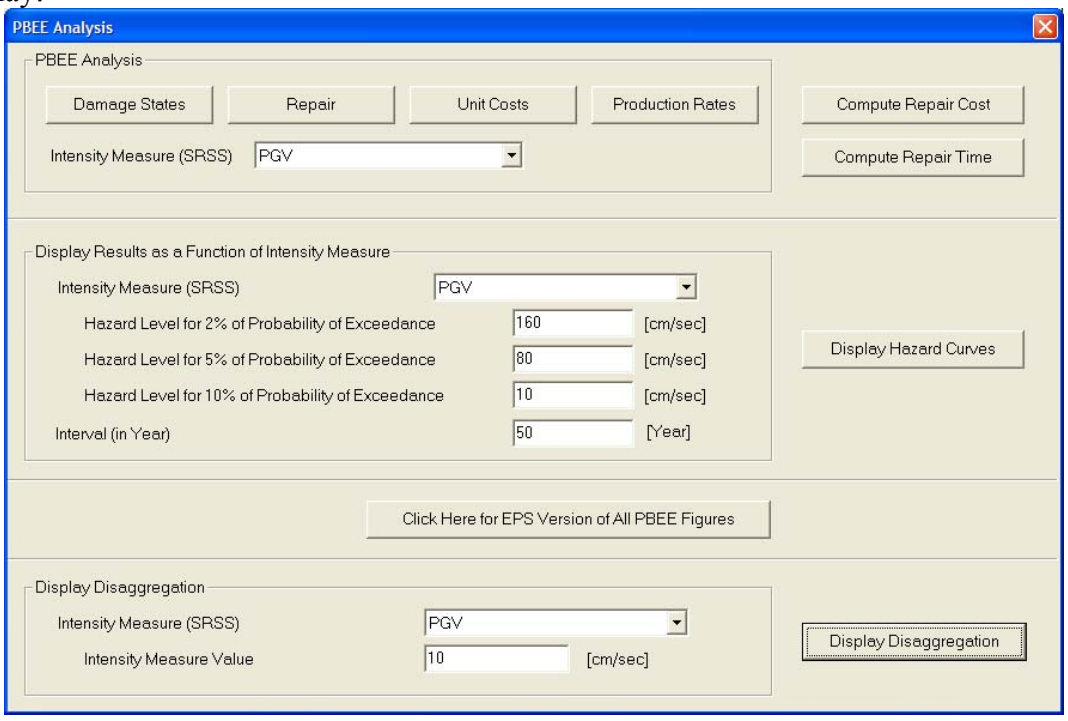


Fig. 57. PBEE analysis window

6.5.1 PBEE Quantities

In the figure above (Fig. 57), only "Damage States" can be currently modified by the user directly within the user interface (however, this is an advanced feature that should be exercised with care, or just left as is). Under Damage States (Fig. 58), Lambda is the median EDP that defines onset of the damage state and is one parameter of the assumed lognormal distribution of damage when conditioned in EDP. It has the same units as the EDP for the selected PG. Beta is the lognormal standard deviation, and is the second parameter of the assumed lognormal distribution. Hence beta is dimensionless and has a typical range between 0 and 1 (although it is not bounded by 1). This parameter is closely related to the coefficient of variation (standard deviation normalized by the mean) under certain conditions (small beta values).

The Repairs, Unit Costs, and Production Rates are displayed in Figs. 59-61, respectively. Users can customize these PBEE quantities through updating a file named PBEE.DLL which is located at the installation folder (C:\Program Files\BridgePBEE or C:\Program Files(x86)\BridgePBEE on a 64bit PC). Please follow the steps described in Appendix E to build an updated PBEE.DLL file and then replace the one in the installation folder.

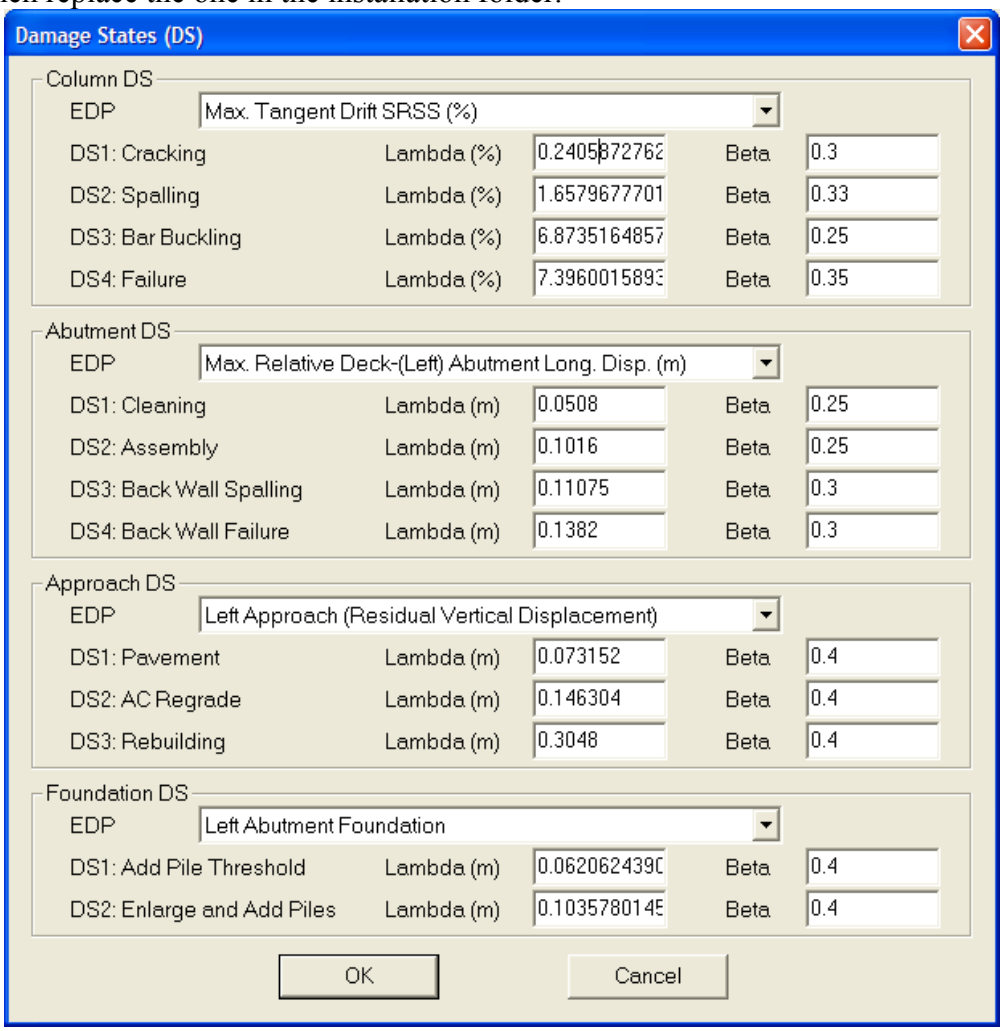
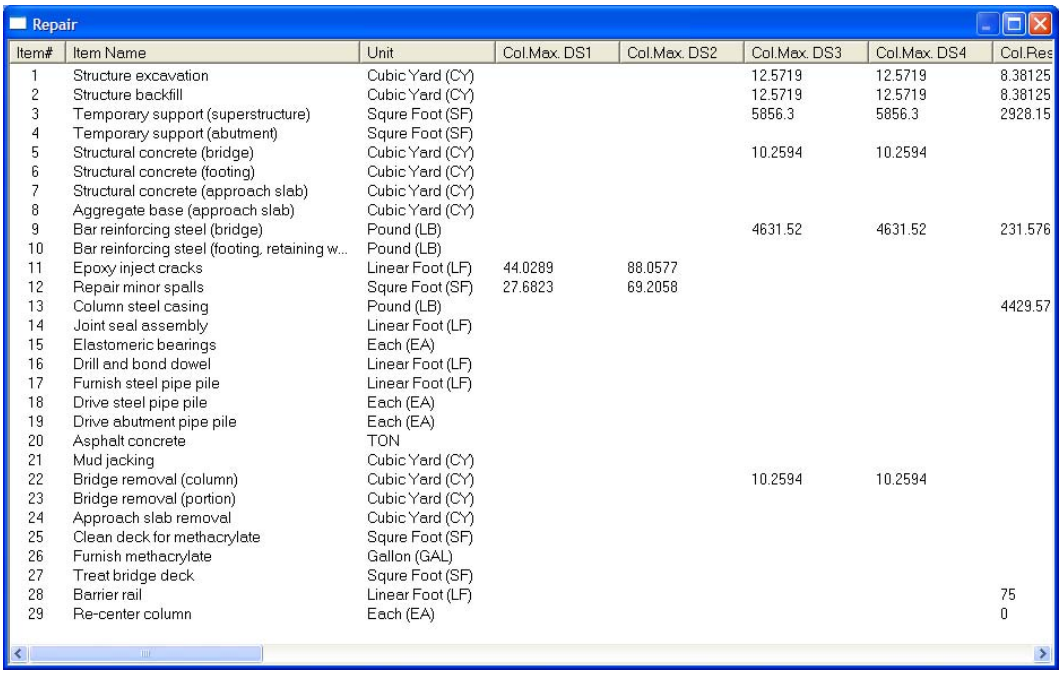


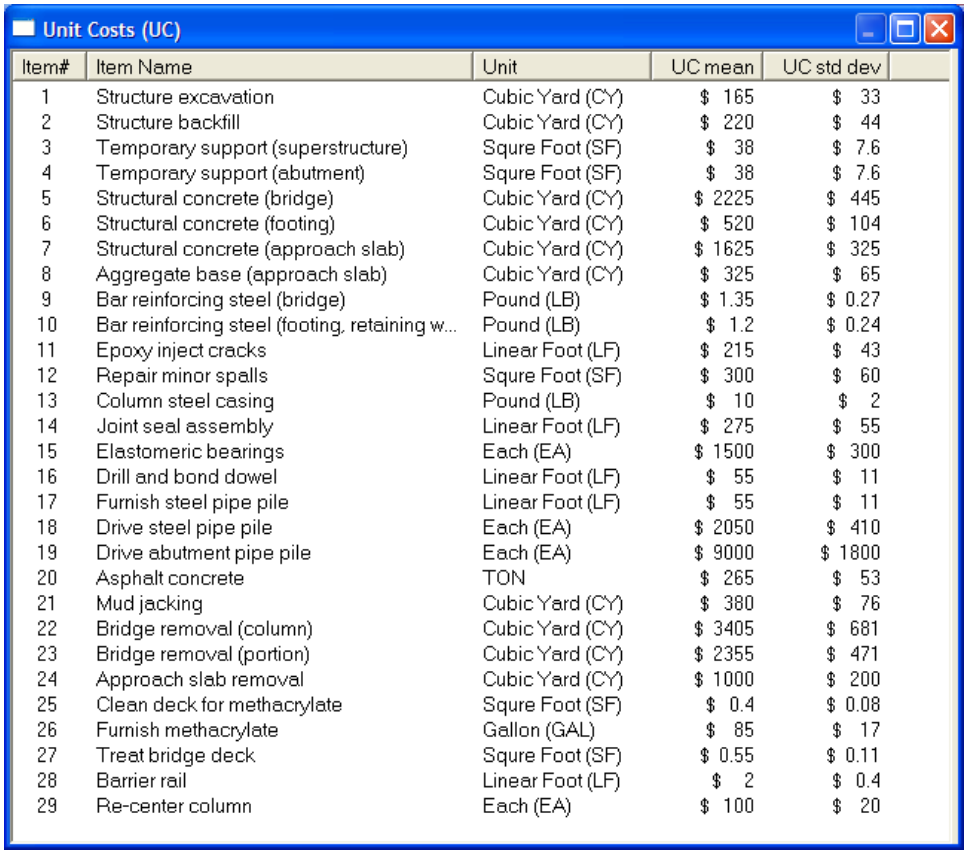
Fig. 58. Damage states window



Repair

Item#	Item Name	Unit	Col.Max. DS1	Col.Max. DS2	Col.Max. DS3	Col.Max. DS4	Col.Res
1	Structure excavation	Cubic Yard (CY)			12.5719	12.5719	8.38125
2	Structure backfill	Cubic Yard (CY)			12.5719	12.5719	8.38125
3	Temporary support (superstructure)	Squre Foot (SF)			5856.3	5856.3	2928.15
4	Temporary support (abutment)	Squre Foot (SF)					
5	Structural concrete (bridge)	Cubic Yard (CY)			10.2594	10.2594	
6	Structural concrete (footing)	Cubic Yard (CY)					
7	Structural concrete (approach slab)	Cubic Yard (CY)					
8	Aggregate base (approach slab)	Cubic Yard (CY)					
9	Bar reinforcing steel (bridge)	Pound (LB)			4631.52	4631.52	231.576
10	Bar reinforcing steel (footing, retaining w...	Pound (LB)					
11	Epoxy inject cracks	Linear Foot (LF)	44.0289	88.0577			
12	Repair minor spalls	Squre Foot (SF)	27.6823	69.2058			
13	Column steel casing	Pound (LB)					4429.57
14	Joint seal assembly	Linear Foot (LF)					
15	Elastomeric bearings	Each (EA)					
16	Drill and bond dowel	Linear Foot (LF)					
17	Furnish steel pipe pile	Linear Foot (LF)					
18	Drive steel pipe pile	Each (EA)					
19	Drive abutment pipe pile	Each (EA)					
20	Asphalt concrete	TON					
21	Mud jacking	Cubic Yard (CY)					
22	Bridge removal (column)	Cubic Yard (CY)			10.2594	10.2594	
23	Bridge removal (portion)	Cubic Yard (CY)					
24	Approach slab removal	Cubic Yard (CY)					
25	Clean deck for methacrylate	Squre Foot (SF)					
26	Furnish methacrylate	Gallon (GAL)					
27	Treat bridge deck	Squre Foot (SF)					
28	Barrier rail	Linear Foot (LF)					75
29	Re-center column	Each (EA)					0

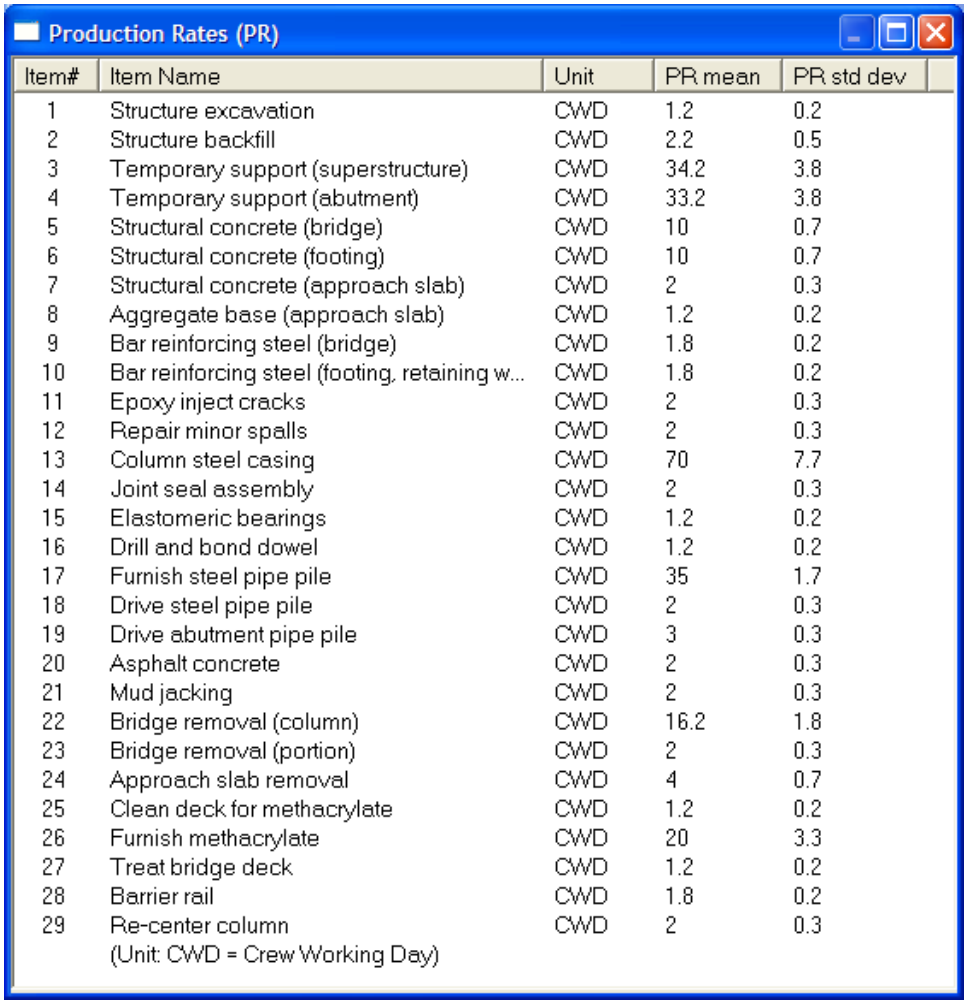
Fig. 59. Repair quantities window



Unit Costs (UC)

Item#	Item Name	Unit	UC mean	UC std dev
1	Structure excavation	Cubic Yard (CY)	$ 165	$ 33
2	Structure backfill	Cubic Yard (CY)	$ 220	$ 44
3	Temporary support (superstructure)	Squre Foot (SF)	$ 38	$ 7.6
4	Temporary support (abutment)	Squre Foot (SF)	$ 38	$ 7.6
5	Structural concrete (bridge)	Cubic Yard (CY)	$ 2225	$ 445
6	Structural concrete (footing)	Cubic Yard (CY)	$ 520	$ 104
7	Structural concrete (approach slab)	Cubic Yard (CY)	$ 1625	$ 325
8	Aggregate base (approach slab)	Cubic Yard (CY)	$ 325	$ 65
9	Bar reinforcing steel (bridge)	Pound (LB)	$ 1.35	$ 0.27
10	Bar reinforcing steel (footing, retaining w...	Pound (LB)	$ 1.2	$ 0.24
11	Epoxy inject cracks	Linear Foot (LF)	$ 215	$ 43
12	Repair minor spalls	Squre Foot (SF)	$ 300	$ 60
13	Column steel casing	Pound (LB)	$ 10	$ 2
14	Joint seal assembly	Linear Foot (LF)	$ 275	$ 55
15	Elastomeric bearings	Each (EA)	$ 1500	$ 300
16	Drill and bond dowel	Linear Foot (LF)	$ 55	$ 11
17	Furnish steel pipe pile	Linear Foot (LF)	$ 55	$ 11
18	Drive steel pipe pile	Each (EA)	$ 2050	$ 410
19	Drive abutment pipe pile	Each (EA)	$ 9000	$ 1800
20	Asphalt concrete	TON	$ 265	$ 53
21	Mud jacking	Cubic Yard (CY)	$ 380	$ 76
22	Bridge removal (column)	Cubic Yard (CY)	$ 3405	$ 681
23	Bridge removal (portion)	Cubic Yard (CY)	$ 2355	$ 471
24	Approach slab removal	Cubic Yard (CY)	$ 1000	$ 200
25	Clean deck for methacrylate	Squre Foot (SF)	$ 0.4	$ 0.08
26	Furnish methacrylate	Gallon (GAL)	$ 85	$ 17
27	Treat bridge deck	Squre Foot (SF)	$ 0.55	$ 0.11
28	Barrier rail	Linear Foot (LF)	$ 2	$ 0.4
29	Re-center column	Each (EA)	$ 100	$ 20

Fig. 60. Unit Costs window



Production Rates (PR)

Item#	Item Name	Unit	PR mean	PR std dev
1	Structure excavation	CWD	1.2	0.2
2	Structure backfill	CWD	2.2	0.5
3	Temporary support (superstructure)	CWD	34.2	3.8
4	Temporary support (abutment)	CWD	33.2	3.8
5	Structural concrete (bridge)	CWD	10	0.7
6	Structural concrete (footing)	CWD	10	0.7
7	Structural concrete (approach slab)	CWD	2	0.3
8	Aggregate base (approach slab)	CWD	1.2	0.2
9	Bar reinforcing steel (bridge)	CWD	1.8	0.2
10	Bar reinforcing steel (footing, retaining w...	CWD	1.8	0.2
11	Epoxy inject cracks	CWD	2	0.3
12	Repair minor spalls	CWD	2	0.3
13	Column steel casing	CWD	70	7.7
14	Joint seal assembly	CWD	2	0.3
15	Elastomeric bearings	CWD	1.2	0.2
16	Drill and bond dowel	CWD	1.2	0.2
17	Furnish steel pipe pile	CWD	35	1.7
18	Drive steel pipe pile	CWD	2	0.3
19	Drive abutment pipe pile	CWD	3	0.3
20	Asphalt concrete	CWD	2	0.3
21	Mud jacking	CWD	2	0.3
22	Bridge removal (column)	CWD	16.2	1.8
23	Bridge removal (portion)	CWD	2	0.3
24	Approach slab removal	CWD	4	0.7
25	Clean deck for methacrylate	CWD	1.2	0.2
26	Furnish methacrylate	CWD	20	3.3
27	Treat bridge deck	CWD	1.2	0.2
28	Barrier rail	CWD	1.8	0.2
29	Re-center column	CWD	2	0.3

(Unit: CWD = Crew Working Day)

Fig. 61. Production Rates window

6.5.2 Compute Repair Cost & Time

Now, you can select any of the Intensity Measures (e.g., PGV above), and then click **Compute Repair Cost** or **Compute Repair Time** in Fig. 57 to display the probabilistic repair cost and Crew Working time in Days (CWD) along with Standard Deviation, displayed for each PG (eleven of them) and each repair quantity (29 of them, see Table 10), as shown below. See Section 7.2.1 for the detailed output.

To convert all PBEE figures to the EPS format, click **Click Here for EPS Version of All PBEE Figures**. A MS Word window with the EPS figures included in the document will pop up once the converting is done (please see Section 7.2.4 for the detailed output).

Table 10. PBEE Repair Quantities

Item#	Item name
1	Structure excavation
2	Structure backfill
3	Temporary support (superstructure)
4	Temporary support (abutment)
5	Structural concrete (bridge)
6	Structural concrete (footing)
7	Structural concrete (approach slab)
8	Aggregate base (approach slab)
9	Bar reinforcing steel (bridge)
10	Bar reinforcing steel (footing, retaining wall)
11	Epoxy inject cracks
12	Repair minor spalls
13	Column steel casing
14	Joint seal assembly
15	Elastomeric bearings
16	Drill and bond dowel
17	Furnish steel pipe pile
18	Drive steel pipe pile
19	Drive abutment pipe pile
20	Asphalt concrete
21	Mud jacking
22	Bridge removal (column)
23	Bridge removal (portion)
24	Approach slab removal
25	Clean deck for methacrylate
26	Furnish methacrylate
27	Treat bridge deck
28	Barrier rail
29	Re-center column

6.5.3 Compute Hazard Curves

The user is also able to specify a Seismic Hazard for a particular geographic location of this bridge system in terms of specified values for any IM (e.g., derived from USGS seismicity maps). The user interface provides default values for site hazard specific to a location in Northern California. The hazard values are provided at each of the 2%-, 5%-, and 10%-probability of exceedance in 50 years only for PGA and PGV. The user should input hazard values specific to the site being studied as well as the intensity measure selected for analysis. If an IM other than PGA or PGV is selected, the user interface will leave the three hazard level input boxes blank for user input as there are no readily available hazard maps or conversions from PGA for an arbitrary IM. The default PGA hazard values were obtained from USGS hazard maps. These PGA values were converted to PGV values using the firm ground conversion of 48 in./sec/g. It is

not meant to imply that switching between PGA and PGV (or any other IM) will yield equal hazard.

Once a desired local site seismicity is defined, users can click **Display Hazard Curves** (Fig. 57) to display the mean annual frequency of exceedance and return period. Please see Section 7.2.2 for the detailed output.

6.5.4 Compute Disaggregation

Users can also click **Display Disaggregation** (Fig. 57) to display the disaggregation by performance groups and repair quantities. Please see Section 7.2.3 for the detailed output. Only the disaggregation of the expected repair cost/time by performance group is possible due to the LLRCAT formulation. However, both expected and variance disaggregation plots are available when disaggregating by repair quantity. The user can select the intensity measure and value on which to disaggregate. The default value is a PGV value equal to the 10% probability of exceedance in 50 years specified in the previous section.

7 Time History and PBEE Output

7.1 Time History Output Quantities

At the end of the FE analysis phase, time histories and bridge responses will be available of the form:

- Column Response Time Histories and Profiles
- Column Response Relationships
- Abutment Responses
- Deformed Mesh

In addition, for PBEE analysis scenarios, Intensity Measures (IMs) and response spectra for each input motion are calculated and are available for display in Table and Figure formats. Performance Group (PG) Quantities and Bridge peak accelerations for all employed shaking motions are also available for display against any of the computed IMs.

The post-processing capabilities can be accessed from Menu **Display** (Fig. 3). Fig. 62 and Fig. 63 show the post-processing capabilities available in a pushover analysis and a base shaking analysis, respectively. Fig. 64 shows the Analysis Options window. Depending on the selection of the Output Data options (Fig. 64), the menu items shown in Fig. 62 and Fig. 63 may be enabled or disabled. For example, In order to view column response profiles and response relationships, **Include Column Response Profiles & Relationships** (Fig. 64) has to be checked before analysis (in this case, menu items of **Column Response Time Histories & Profiles** as well as **Column Response Relationships** shown in Fig. 62 and Fig. 63 will be enabled). To view the deformed mesh (and animation), both Output Data options of **Include Column Response Profiles & Relationships** and **Include Soil Displacement** (Fig. 64) must be checked.

If the user wants to view the deformed mesh for the final step only, check **Display Deformed Mesh for Final Step Only (Recommended for Large Models)** (Fig. 64). The option is particularly useful when the output data is large and all output cannot be loaded into memory.

To display output for a different input motion, please follow steps shown in Fig. 65. The name of the selected input motion will also appear on the menu items (Fig. 63).

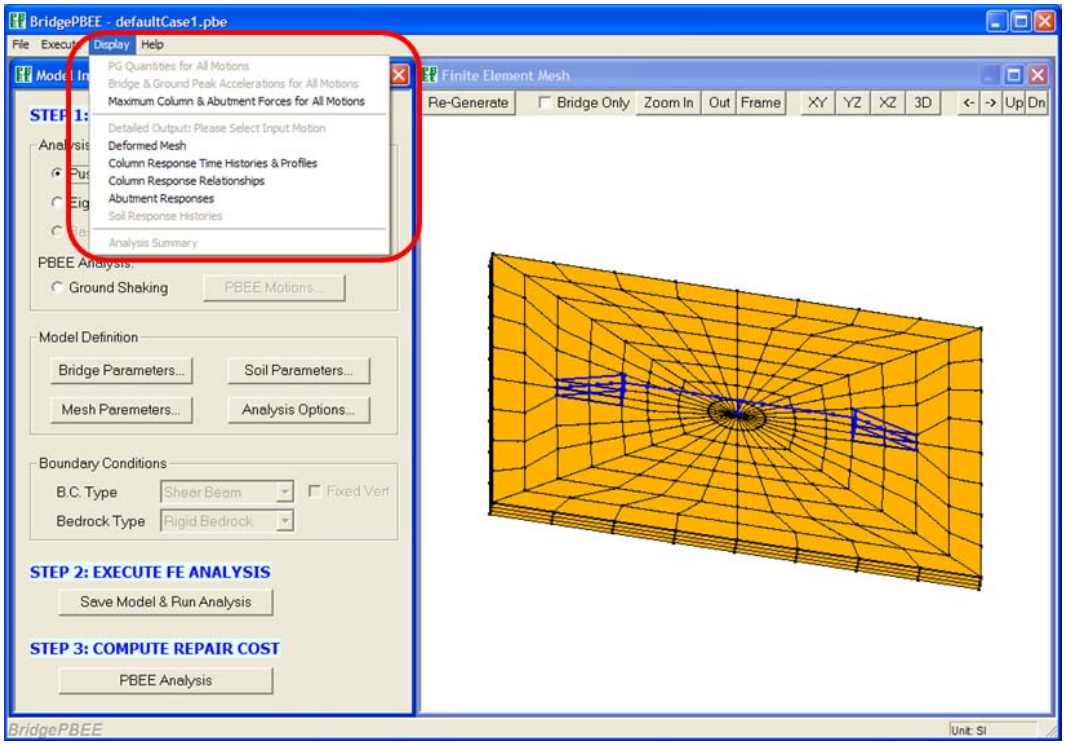


Fig. 62. Post-processing capabilities (menu options) available in a pushover analysis

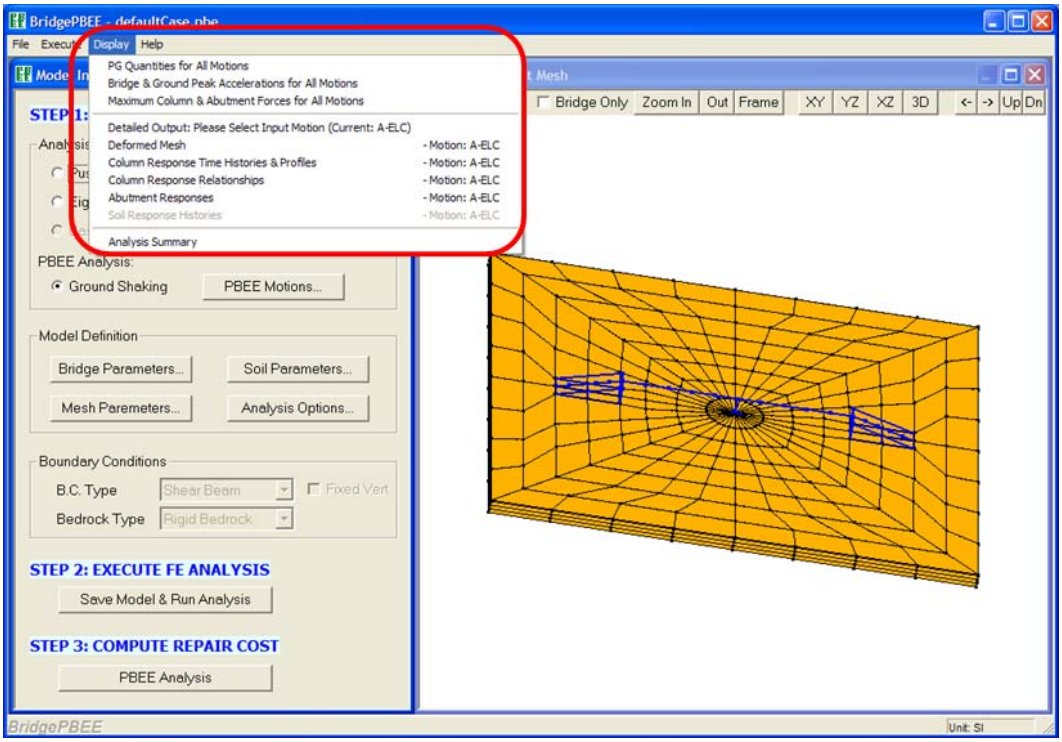


Fig. 63. Post-processing capabilities (menu options) available in a base shaking analysis

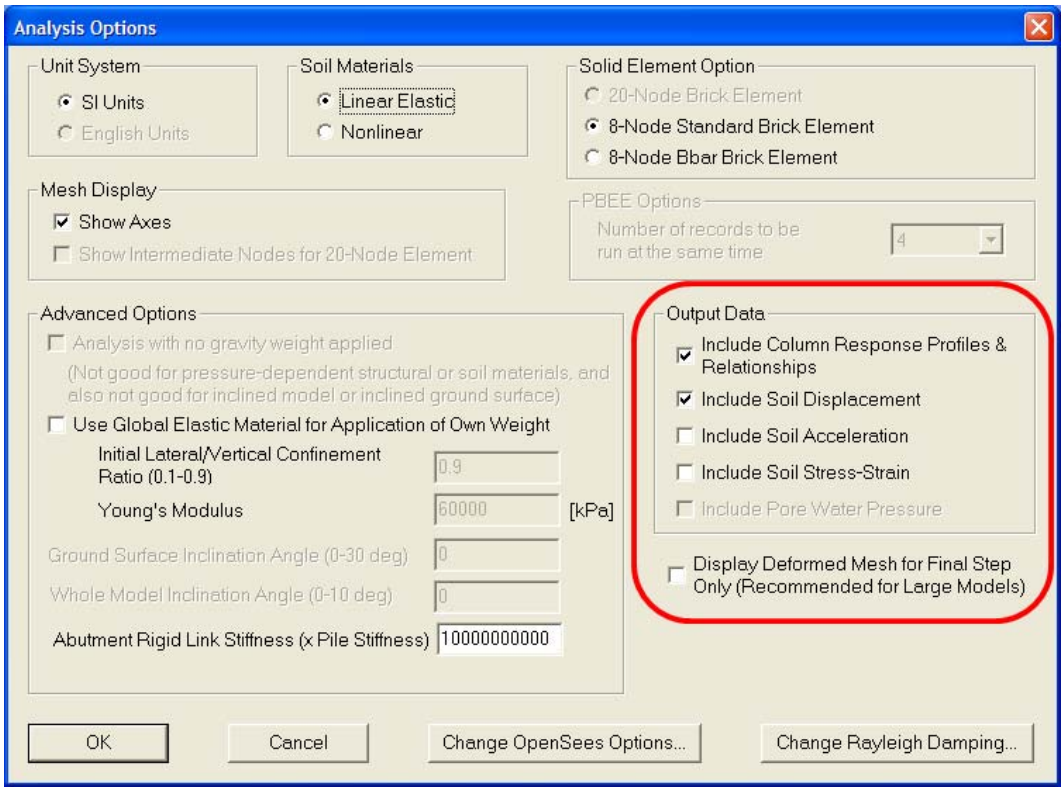
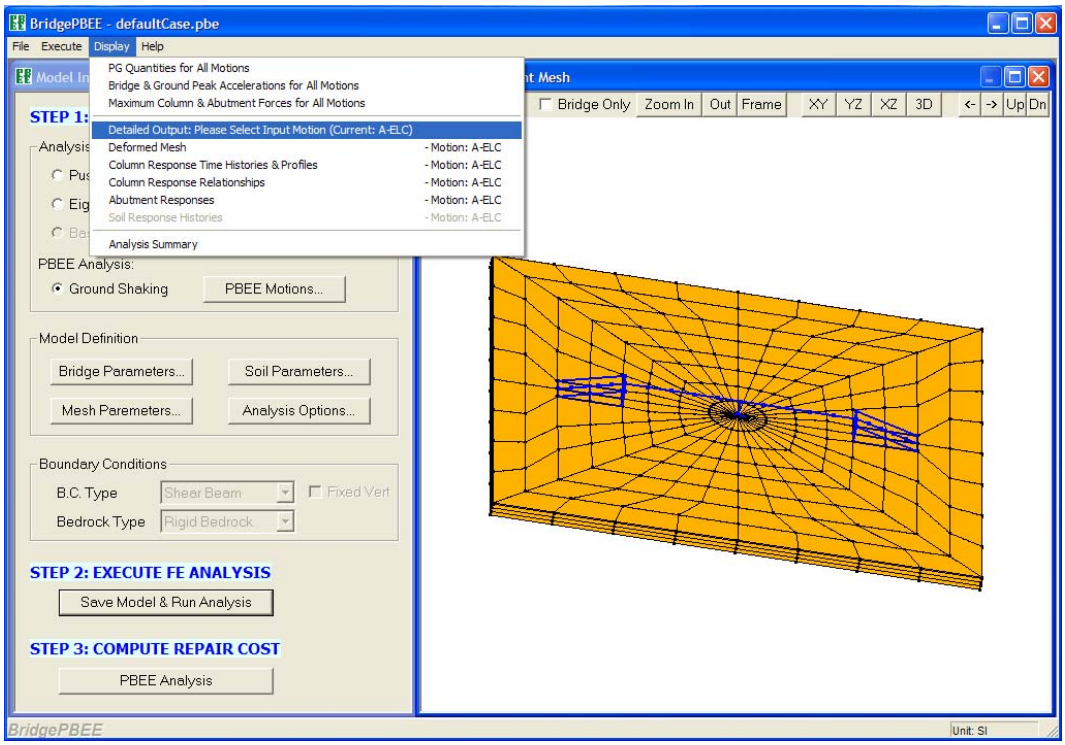
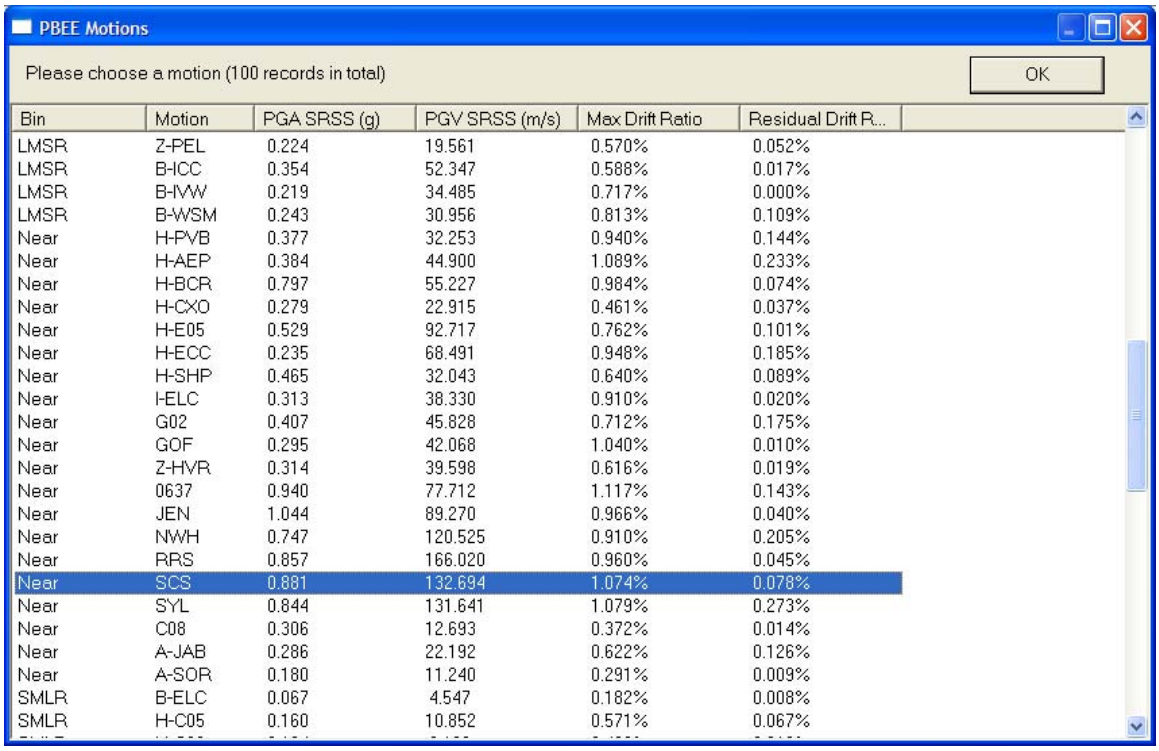


Fig. 64. Analysis options in BridgePBEE



a)



Bin	Motion	PGA SRSS (g)	PGV SRSS (m/s)	Max Drift Ratio	Residual Drift R...
LMSR	Z-PEL	0.224	19.561	0.570%	0.052%
LMSR	B-ICC	0.354	52.347	0.588%	0.017%
LMSR	B-IVW	0.219	34.485	0.717%	0.000%
LMSR	B-WSM	0.243	30.956	0.813%	0.109%
Near	H-PVB	0.377	32.253	0.940%	0.144%
Near	H-AEP	0.384	44.900	1.089%	0.233%
Near	H-BCR	0.797	55.227	0.984%	0.074%
Near	H-CXO	0.279	22.915	0.461%	0.037%
Near	H-E05	0.529	92.717	0.762%	0.101%
Near	H-ECC	0.235	68.491	0.948%	0.185%
Near	H-SHP	0.465	32.043	0.640%	0.089%
Near	I-ELC	0.313	38.330	0.910%	0.020%
Near	G02	0.407	45.828	0.712%	0.175%
Near	GOF	0.295	42.068	1.040%	0.010%
Near	Z-HVR	0.314	39.598	0.616%	0.019%
Near	0637	0.940	77.712	1.117%	0.143%
Near	JEN	1.044	89.270	0.966%	0.040%
Near	NWH	0.747	120.525	0.910%	0.205%
Near	RRS	0.857	166.020	0.960%	0.045%
Near	SCS	0.881	132.694	1.074%	0.078%
Near	SYL	0.844	131.641	1.079%	0.273%
Near	C08	0.306	12.693	0.372%	0.014%
Near	A-JAB	0.286	22.192	0.622%	0.126%
Near	A-SOR	0.180	11.240	0.291%	0.009%
SMLR	B-ELC	0.067	4.547	0.182%	0.008%
SMLR	H-C05	0.160	10.852	0.571%	0.067%

b)

Fig. 65. Steps to display output for a different input motion: a) click menu Display (Fig. 3); b) select an input motion

7.1.1 Column Response Time Histories and Profiles

The column response time histories and response profiles can be accessed by clicking menu **Display** (Fig. 3) and then **Column Response Time Histories and Profiles** (Fig. 66). The column response window is shown in Fig. 67. There are 3 dropdown lists available for users to choose. The contents of the 3 lists are as follows:

Left Dropdown List:

- **Response Histories**
- **Response Profiles**

Middle Dropdown List:

- **Displacement** *(relative to base of soil mesh for earthquake excitation scenarios)*
- **Acceleration** *(Absolute or Total)*
- **Rotation**
- **Bending Moment**
- **Shear Force**
- **Pressure**
- **Response Summary**

Right Dropdown List:

- **Longitudinal Direction**

- **Transverse Direction**
- **Vertical Direction**

Please note that the above Middle Dropdown List is only valid for the longitudinal and transverse directions. If the Vertical Direction in the Right Dropdown List is selected, the Middle Dropdown List will become (the displacement refers to the one relative to the model base):

- **Displacement**
- **Acceleration**
- **Rotation**
- **Torsional Moment (Torque)**
- **Axial Force**

1) Column Response Profiles

The column response profile will be displayed if **Response Profiles** in the Left Dropdown List (Fig. 67) is selected. For example, Fig. 68 shows the bending moment in the longitudinal plane. The horizontal axis of the plot is the response name (e.g., displacement, bending moment, etc.) and the vertical axis is the elevation of the column (and the pile shaft below grade). Zero elevation means the ground surface.

For **Displacement**, **Acceleration** and **Rotation**, two lines are plotted for the response profile selected (these lines are continuous):

- **End**: the response profile at the final step
- **Max**: the response profile at a certain step when the maximum (absolute) value occurs

In the cases of **Bending Moment**, **Shear Force**, and **Pressure**, three lines are plotted:

- **End (Envelope)**: Envelope of the response values at the final step
- **End (Element Output)**: the response values for both nodes (top node/bottom node) of every element (this line is discontinuous)
- **Max**: the response profile at a certain step when the maximum (absolute) value occurs

Pressure is the difference of the shear forces at the both ends of each element divided by the element length.

If **Response Summary** is selected (the Middle Dropdown List, see Fig. 67), response profiles of displacement, bending moment, shear force and pressure will be plotted in one window (Fig. 69).

To convert all the plots in the current window to a PDF file (Adobe PDF or similar has to be available) or send to a printer, click **Print** located near the top-left corner of the window (this feature is available in all figure windows in BridgePBEE). To view the data of each plot (this feature is also available in all plots in BridgePBEE), click the filename (e.g., click momProf.txt in Fig. 68 and an Internet Explorer window will pop up and display the momProf.txt data file).

2) Column Response Time Histories

The column response time history will be displayed if **Response Time Histories** in the Left Dropdown List (Fig. 67) is selected. Fig. 70 shows the window for displaying the column longitudinal displacement time histories. The top plot in the window (Fig. 70a) is the response profiles for at specific load steps, while the remaining plots are the response time histories at different depths (Fig. 70b).

In the plot for the **Response Profiles for All Steps** (Fig. 70a), only 20 steps including the initial state (Step 0), the first step and the final step are shown if more than 20 steps are simulated. Step 0 refers to the initial state after application of own weight and before the dynamic run (i.e., pushover or earthquake shaking).

If **Acceleration** is selected, the free-field acceleration response time history and the input acceleration time history are also plotted (Fig. 71). The free-field location is shown in Fig. 66 (at the ground surface along the diagonal line of the mesh near the edge corner node).

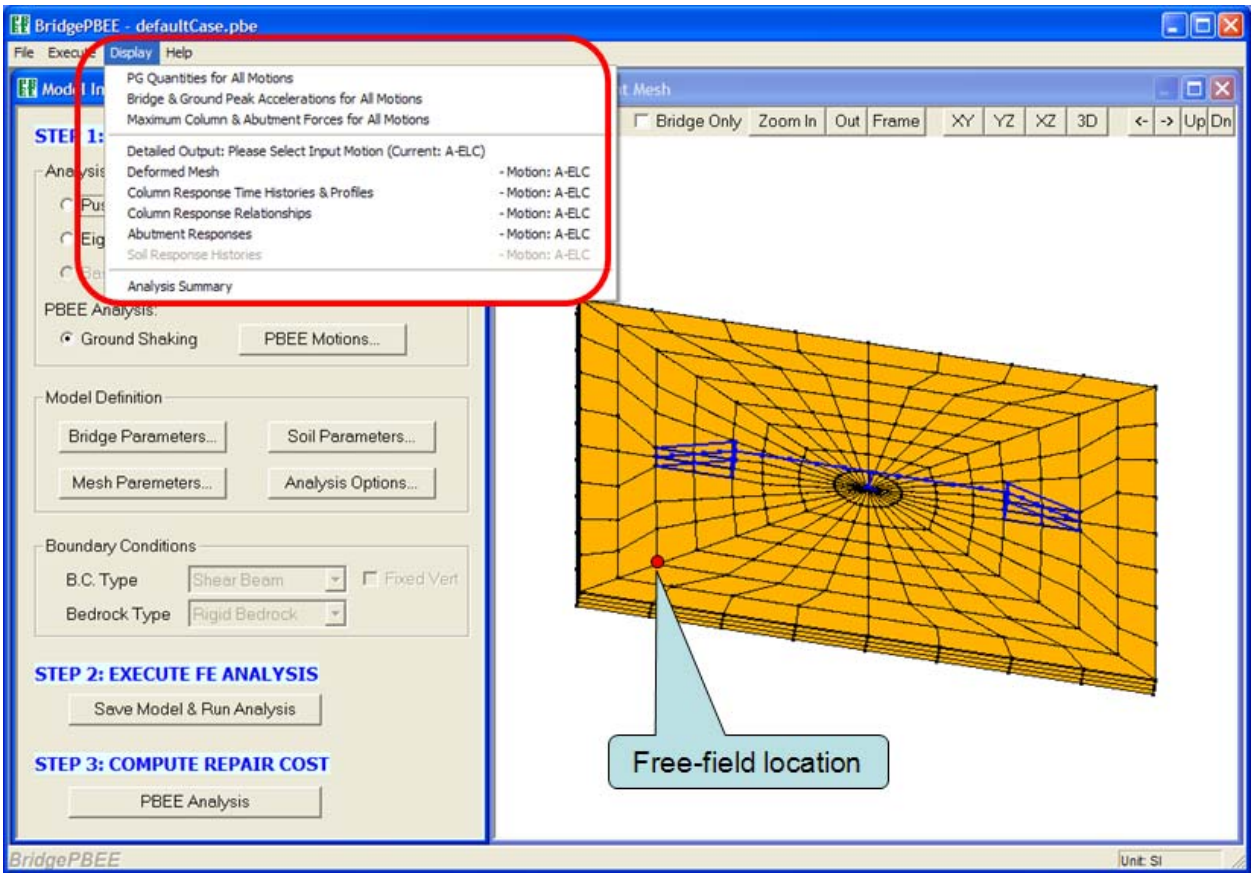


Fig. 66. Menu items to access the column response time histories and response profiles

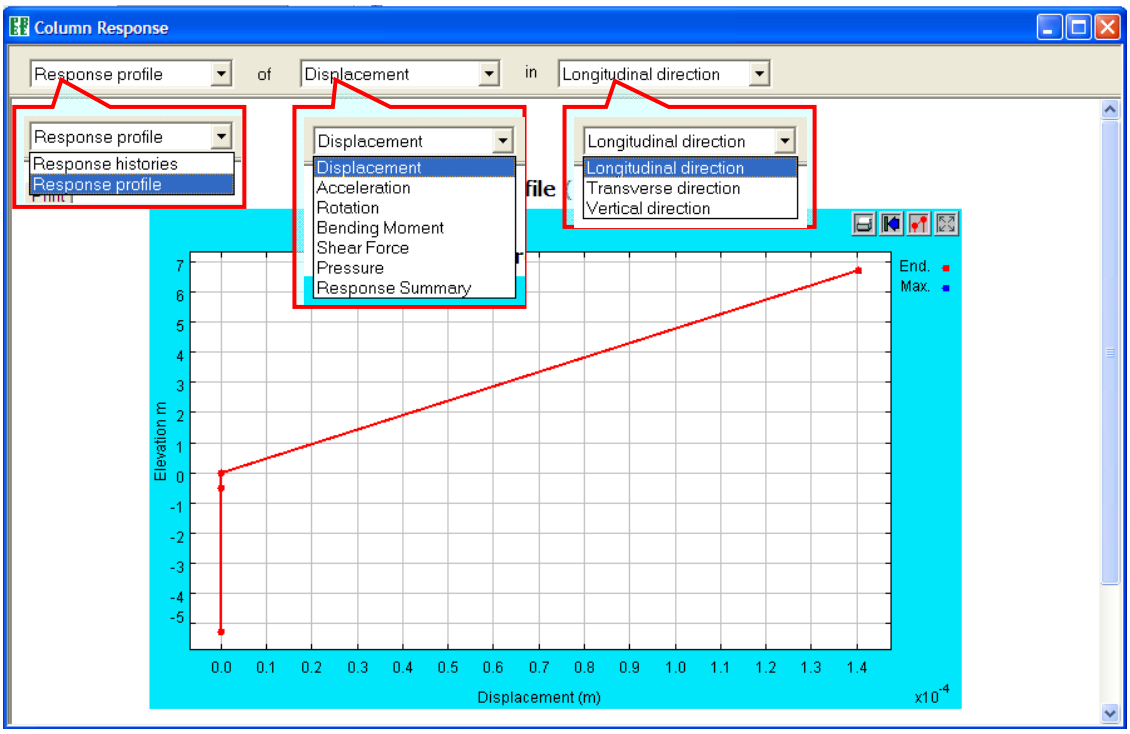


Fig. 67. Response time histories and profiles for column (and pile shaft): displacement is shown at the nodes (only one element is used above ground).

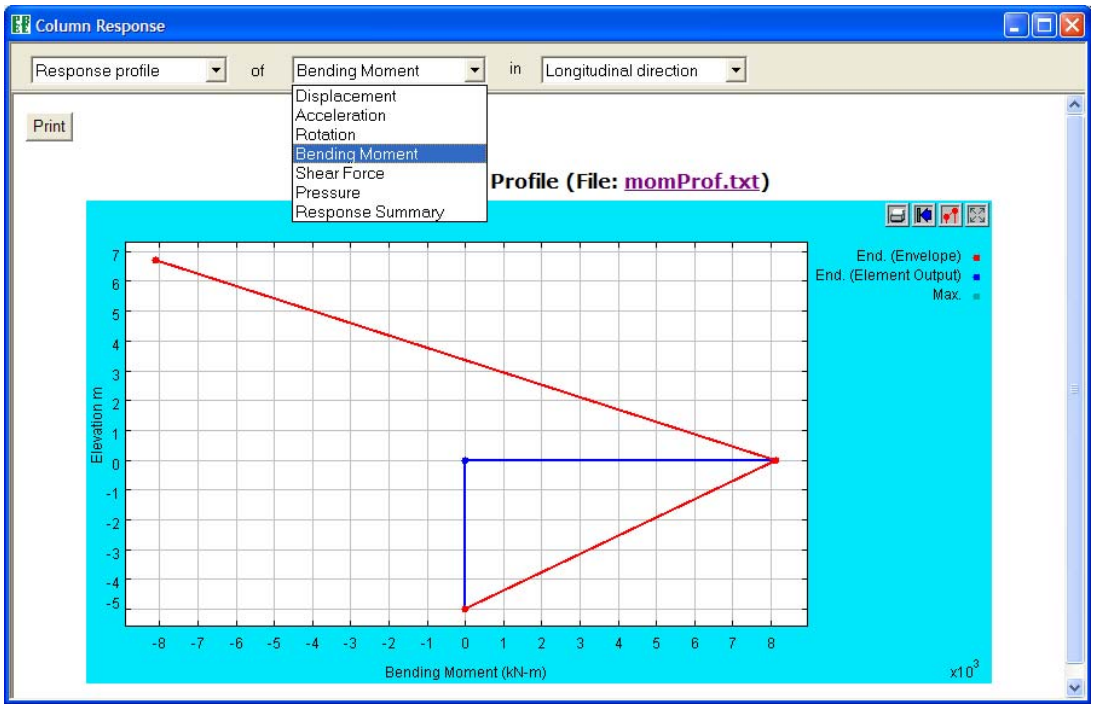


Fig. 68. Bending moment profile in the longitudinal plane

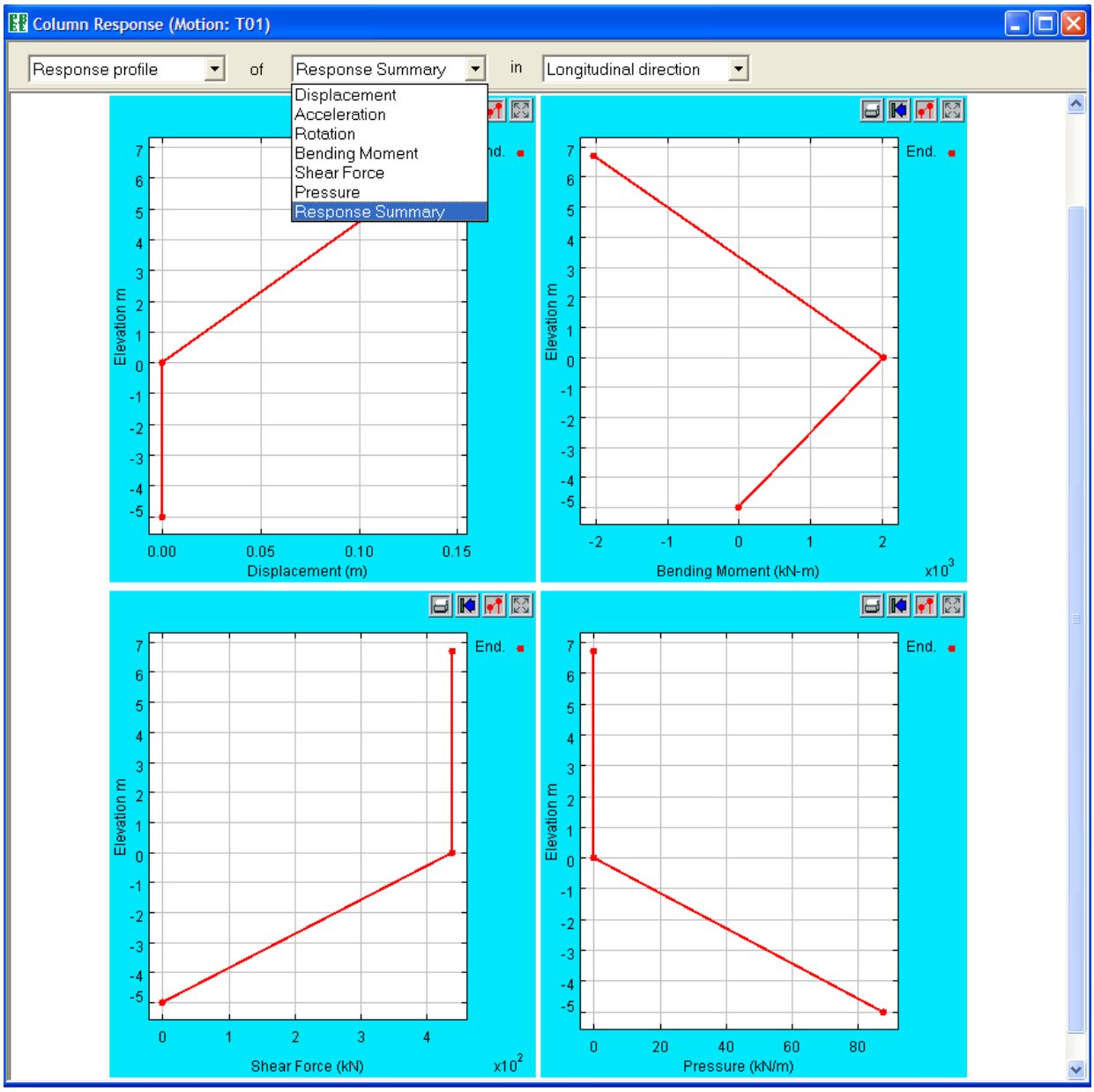
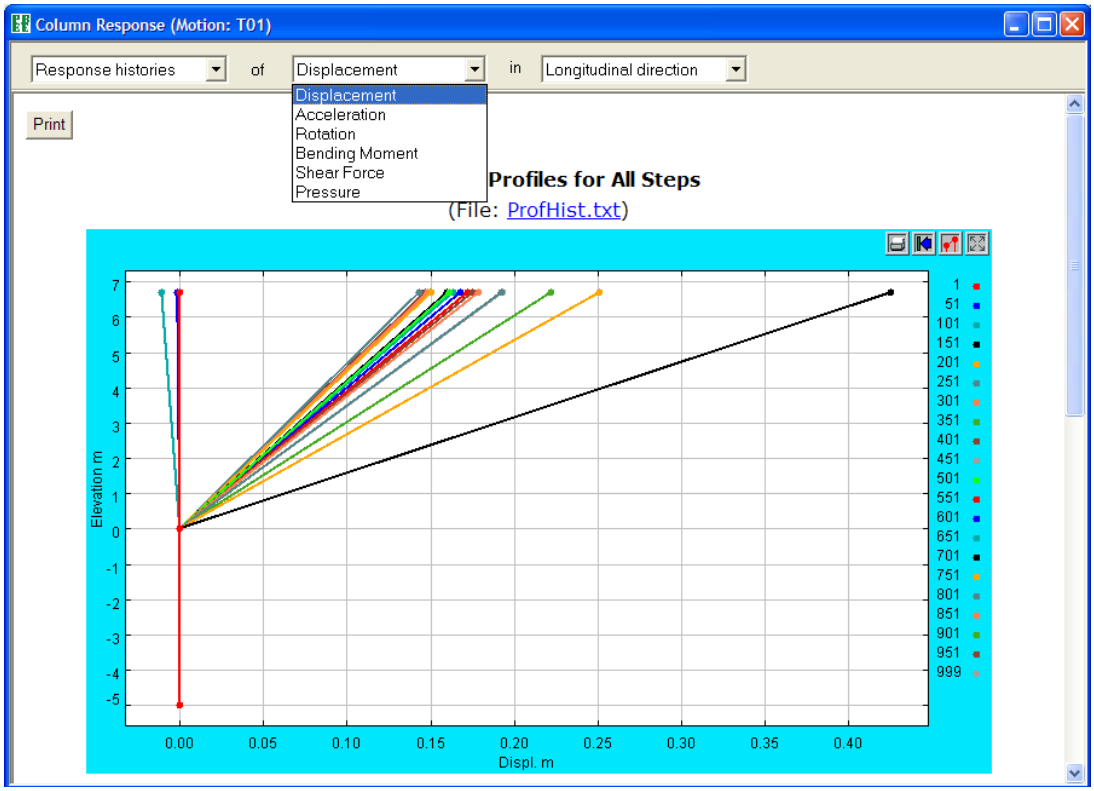
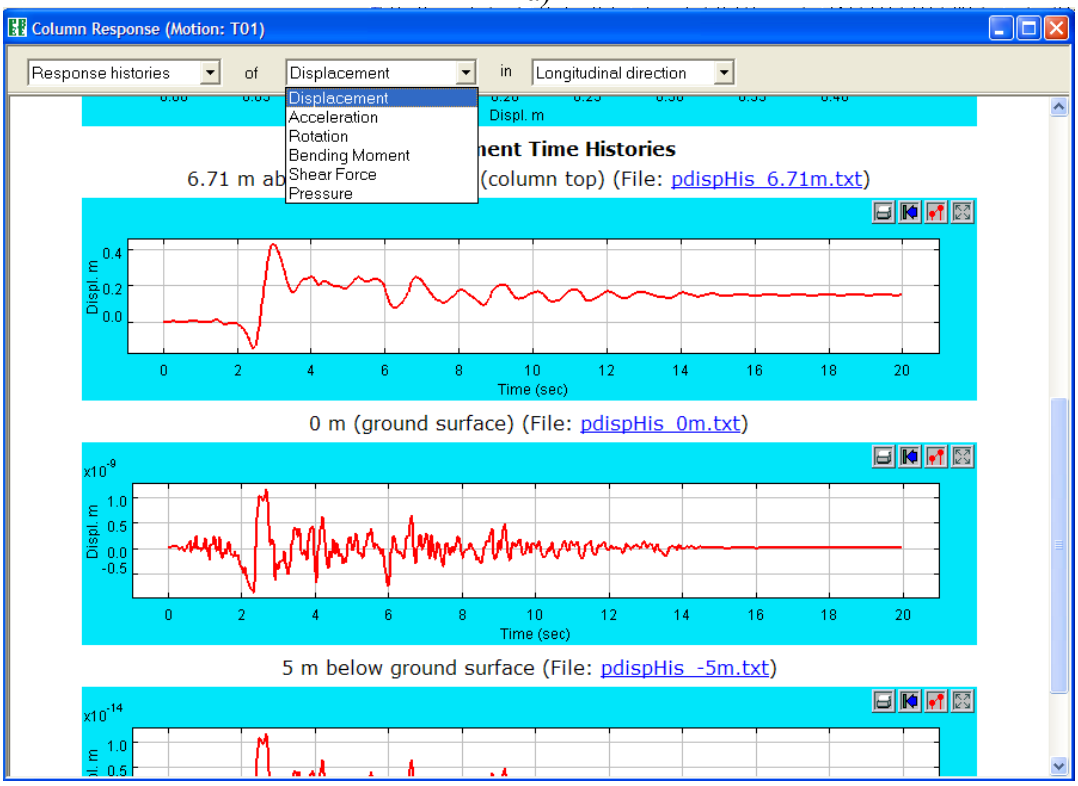


Fig. 69. Response summary

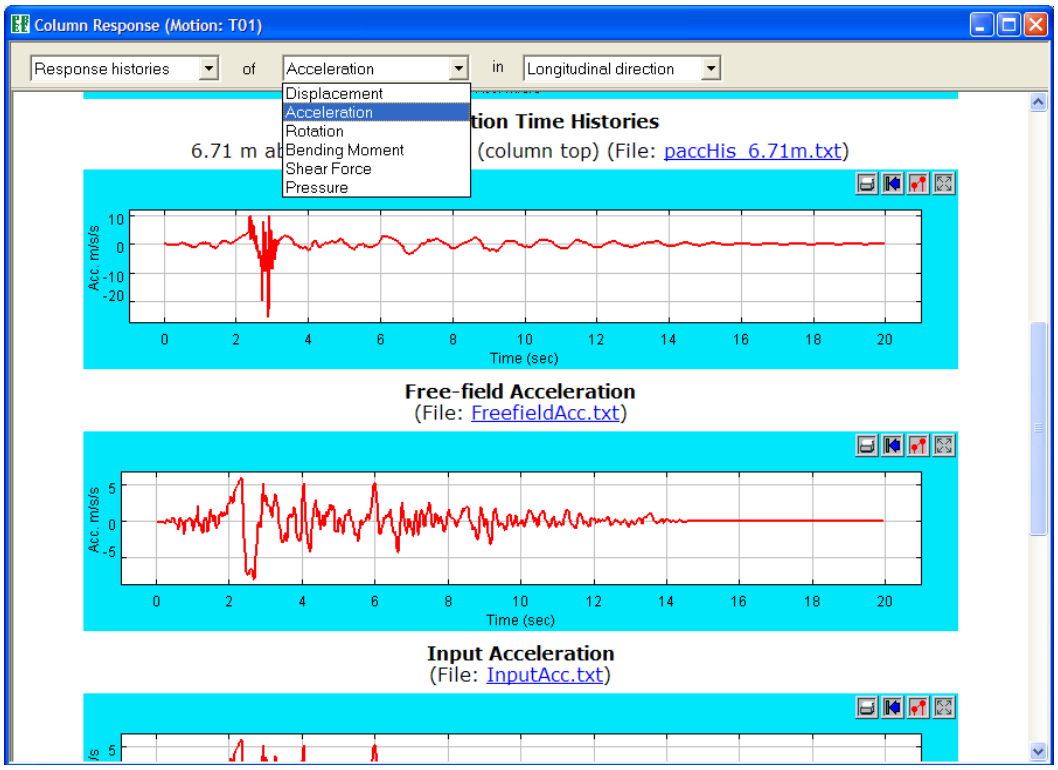


a)



b)

Fig. 70. Column longitudinal displacement response time histories: a) response profiles at specific load steps; b) response time histories at different elevations



b)

Fig. 71. Column longitudinal acceleration response time histories at different elevations (free-field and input accelerations are also included)

7.1.2 Column Response Relationships

The column response relationships can be accessed by clicking menu **Display** (Fig. 3) and then **Column Response Relationships** (Fig. 72). The column response relationships window is shown in Fig. 73. There are 3 dropdown lists available for users to choose from. The contents of the 3 lists are as follows:

Left Dropdown List:

- **Load-displacement**
- **Moment-curvature**

Right Dropdown List:

- **Longitudinal Direction**
- **Transverse Direction**

The Middle Dropdown List includes all elevations (starting from column top). Again, zero elevation refers to the ground surface.

Fig. 74 shows the longitudinal load-displacement curve at the column top. The load refers to the shear force of the beam-column element at the specified elevation. Fig. 75 shows the moment-curvature curve at the column top. The vertical axis is the bending moment and the horizontal

axis is the curvature. To view the data for the plot, click the .txt filename (e.g., click curvX_6.71m.txt in Fig. 75).

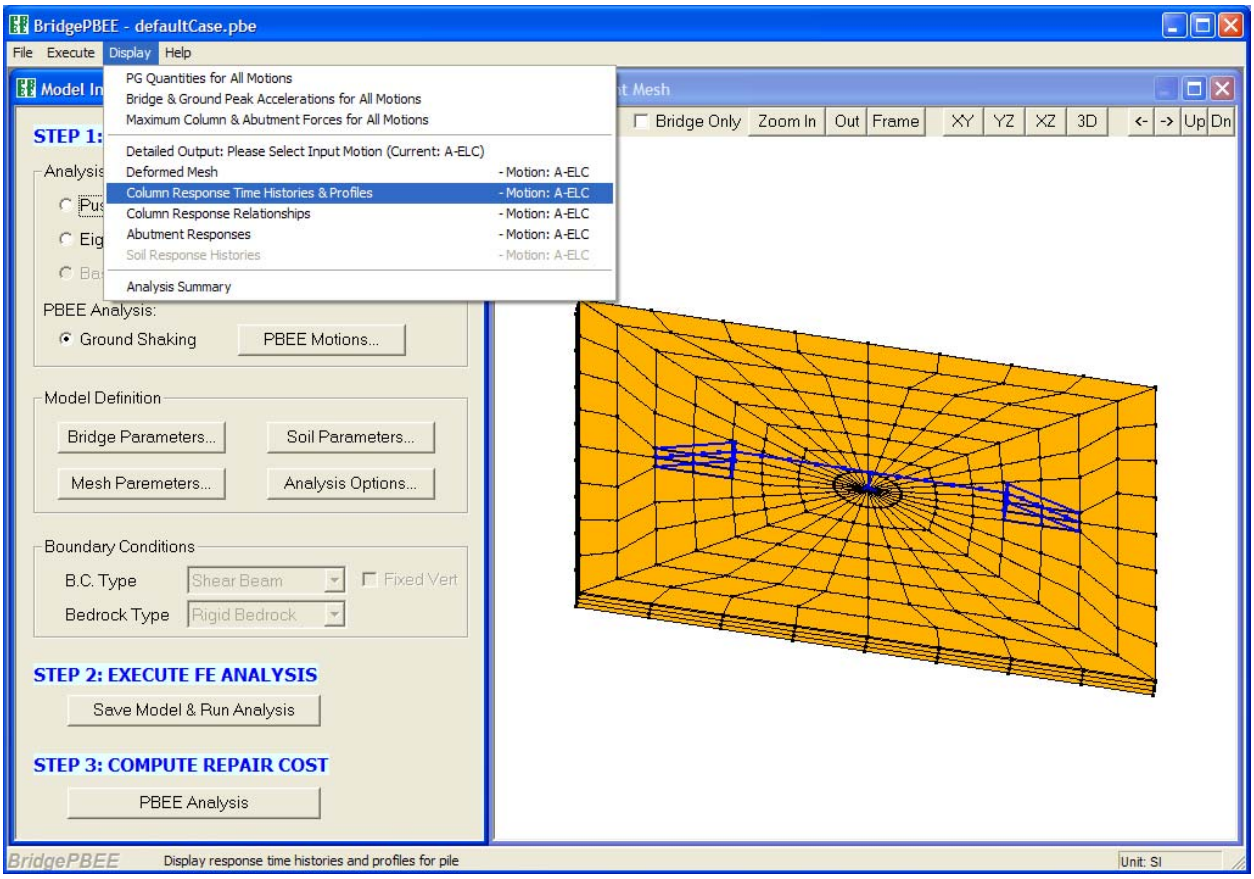


Fig. 72. Menu items to access the column response relationships

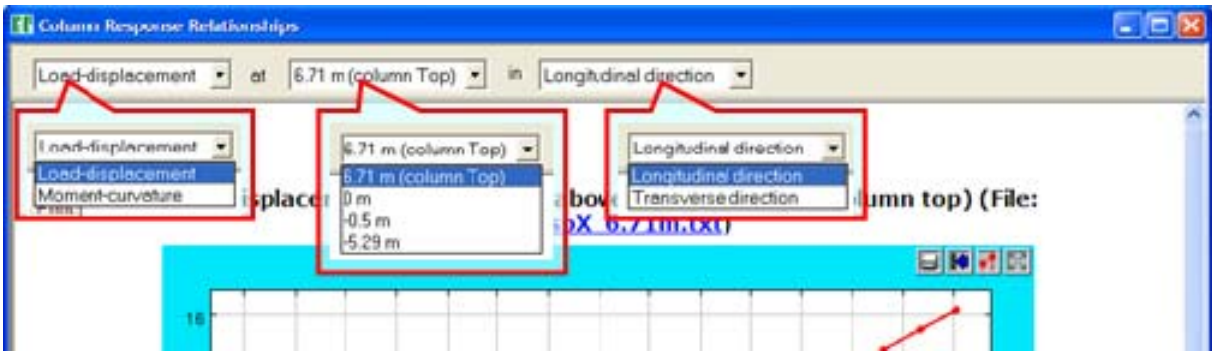


Fig. 73. Column response output options

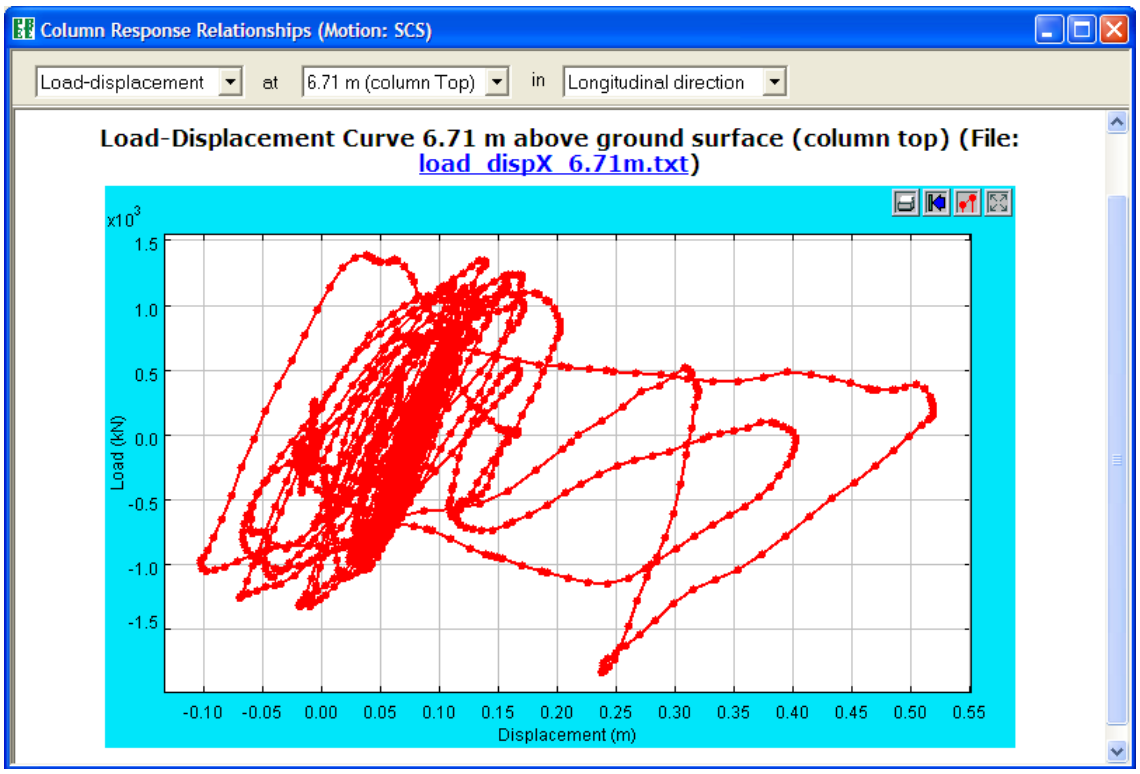


Fig. 74. Load-displacement curve at column top

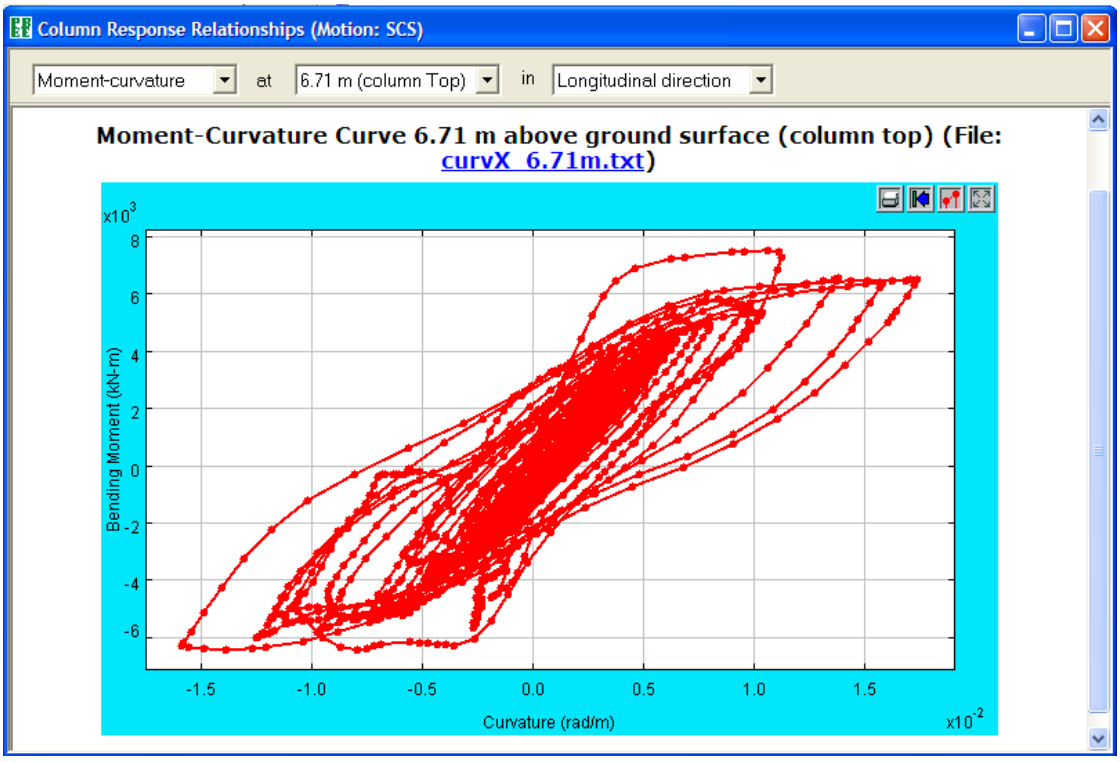


Fig. 75. Moment-curvature curve at column top

7.1.3 Abutment Responses Time Histories

The abutment responses can be accessed by clicking menu **Display** and then **Abutment Responses** (Fig. 76). The abutment responses window includes the following options (Fig. 77):

- **Force-Displacement Relationships**
- **Relative Deck-end/Abutment Displacement Time Histories**
- **Resisting Force Time Histories**
- **Pile Cap Displacement Time Histories**

where Pile Cap refers to the embankment base right below the deck-end (please see Fig. 86 in Section 7.1.6).

Three directions (longitudinal, transverse and vertical directions) of the above responses for both left and right abutments are all displayed. Fig. 78 shows the abutment response time histories. The force refers to the resisting force acting on deck-end and the displacement refers to the relative deck-end/abutment displacement.

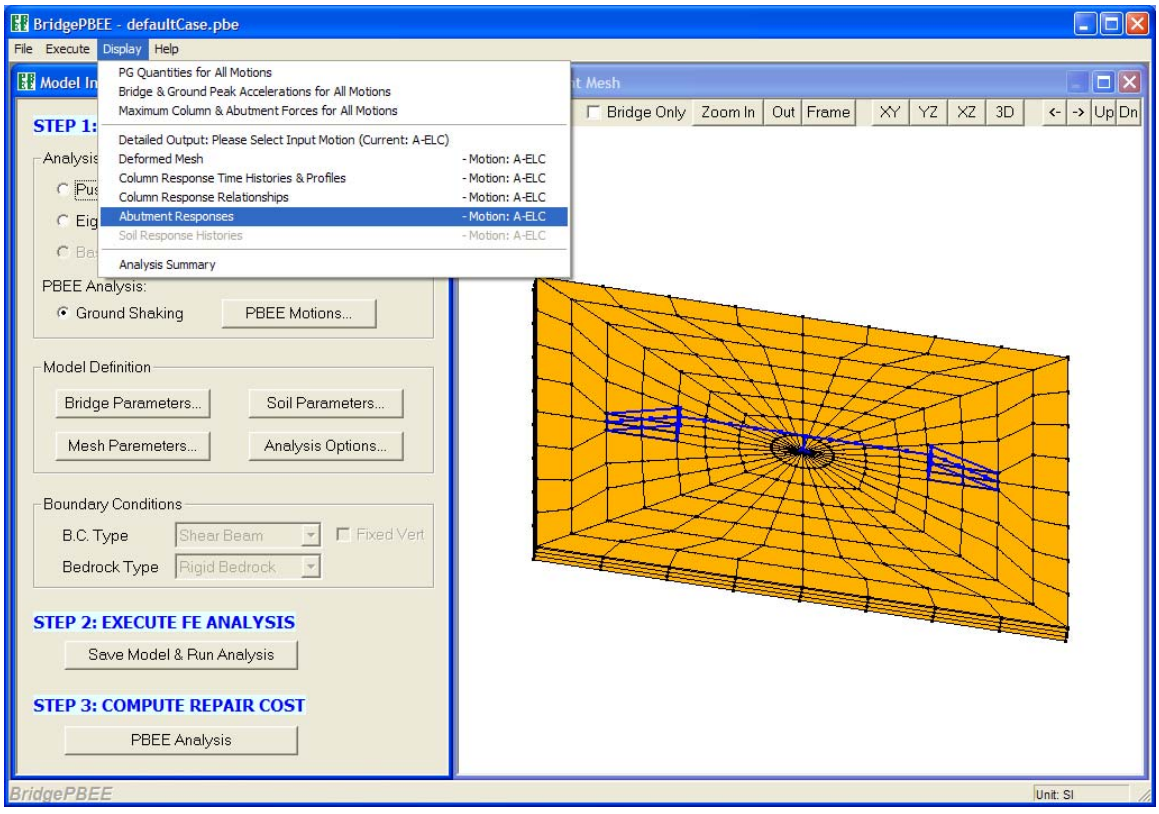


Fig. 76. Menu items to access the abutment responses

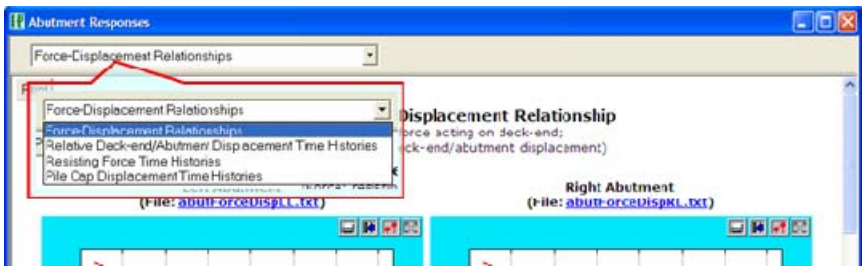
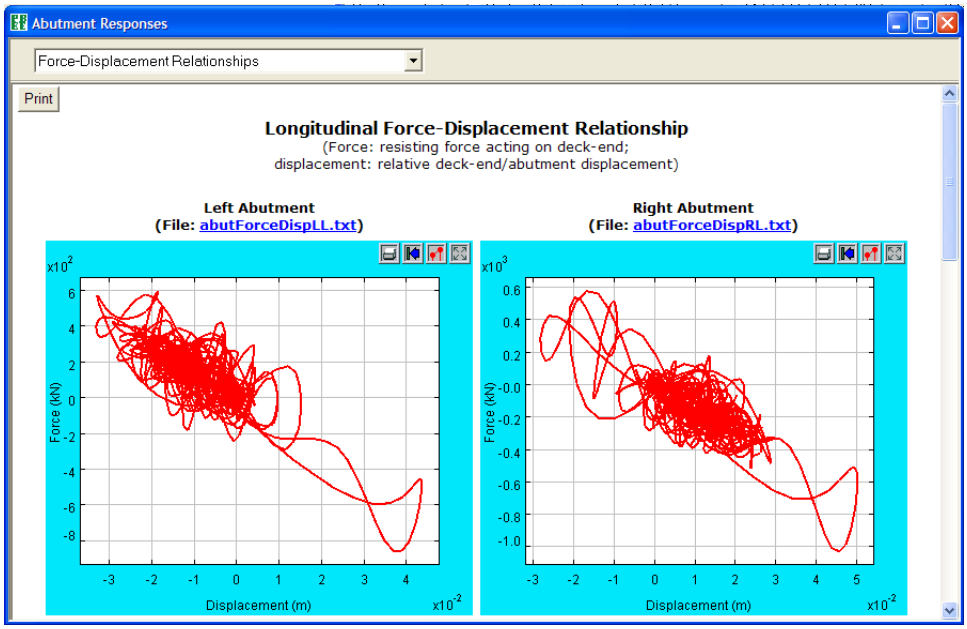
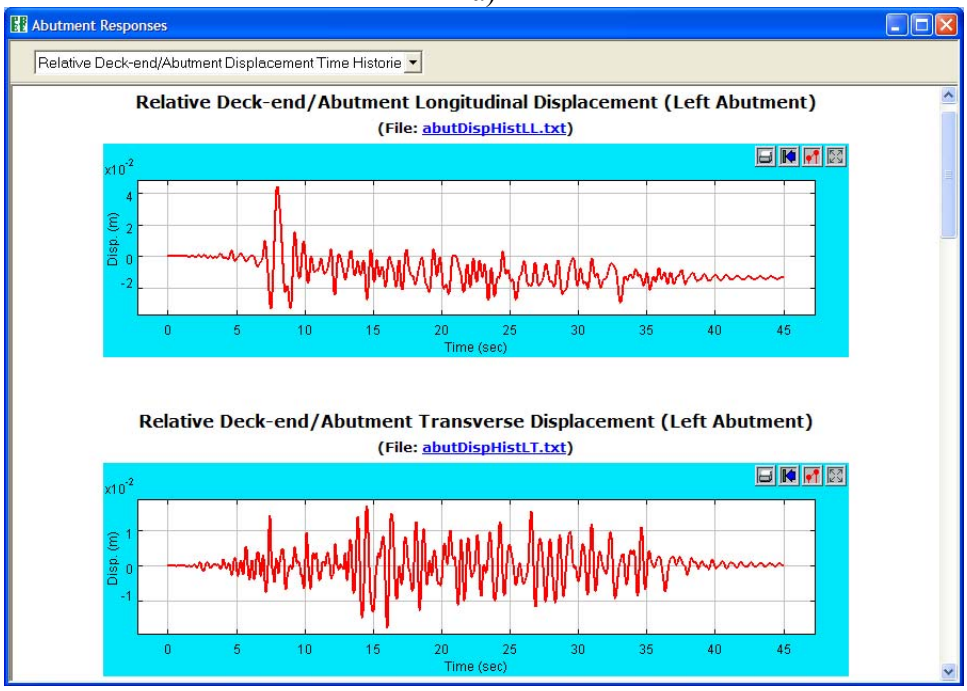


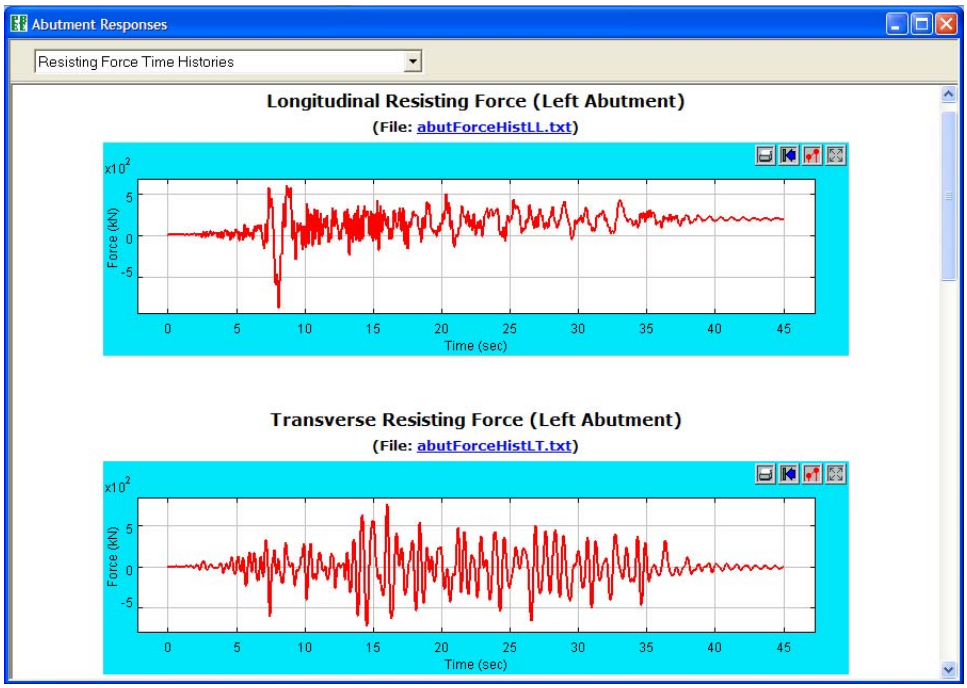
Fig. 77. Menu items to access the abutment responses



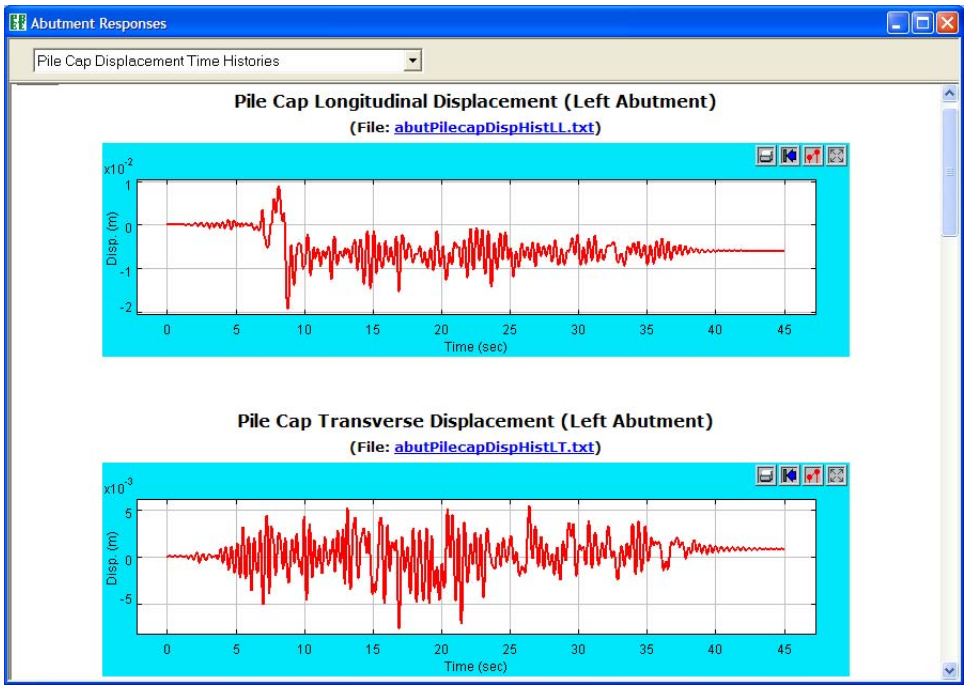
a)



b)



c)



d)

Fig. 78. Abutment response time histories (scroll down to see all directions): a) abutment force-displacement relationships; b) relative deck-end/abutment displacement time histories; c) resisting force time histories; and d) abutment pile cap time histories

7.1.4 Deformed Mesh

The deformed mesh can be accessed by clicking menu **Display** (Fig. 3) and then **Deformed Mesh** (Fig. 79). The deformed mesh window is shown in Fig. 80. There are 3 dropdown lists available for users to choose. The contents of the 3 lists are as follows:

Left Dropdown List:

- **Due to gravity (soil only)**
- **Due to gravity (bridge included)**
- **Due to pushover** (or **Due to base shaking**)

Middle Dropdown List:

- **Deformed mesh**
- **Disp. contour fill**
- **X-Disp contour**
- **Y-Disp contour**
- **Z-Disp contour**

The Right Dropdown List includes options of 3D view as well as 2D views for a number of pre-defined planes.

To view the bridge structure only, check **Bridge Only** in the bottom-right corner of the window (Fig. 80).

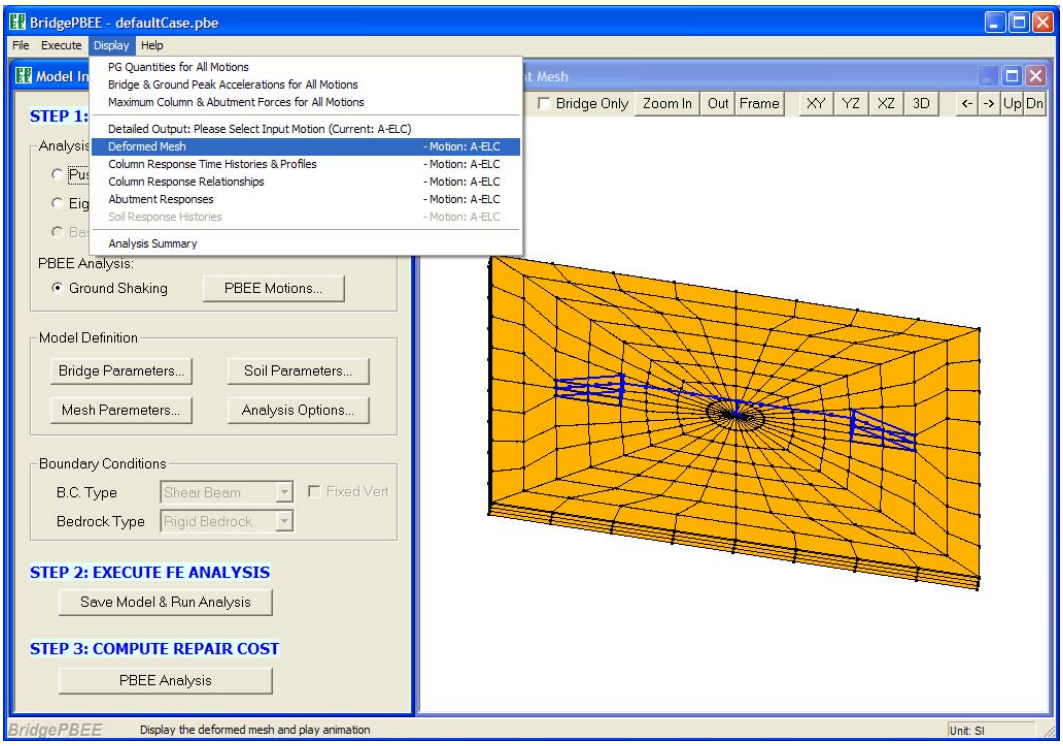


Fig. 79. Menu items to access the deformed mesh

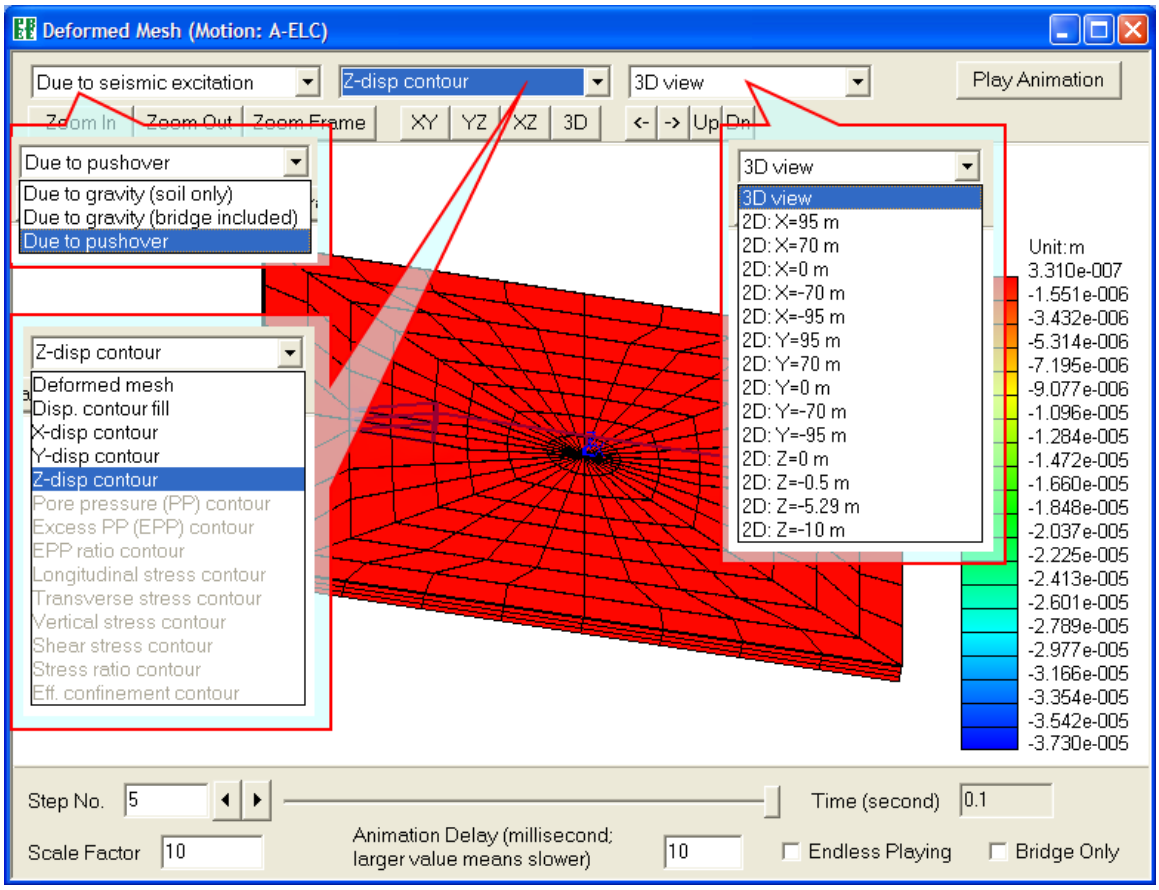


Fig. 80. Deformed mesh

7.1.5 Soil Response Time Histories

The soil response time histories can be accessed by clicking menu **Display** (Fig. 3) and then **Soil Response Histories** (Fig. 81). The soil response window is shown in Fig. 82. There are 3 dropdown lists available for users to choose. The contents of the 3 lists are as follows:

Left Dropdown List (Fig. 82):

- **Longitudinal acceleration time histories**
- **Longitudinal displacement (rel. to base) histories**
- **Transverse acceleration time histories**
- **Transverse displacement (rel. to base) histories**
- **Vertical acceleration time histories**
- **Vertical displacement time histories**
- **Excess pore pressure time histories**
- **Shear stress (zx) vs. strain & eff. confinement**
- **Shear stress (yz) vs. strain & eff. confinement**
- **Longitudinal normal stress time histories**
- **Transverse normal stress time histories**

- **Shear stress (zx) time histories**
- **Shear stress (yz) time histories**

Right Dropdown List (Fig. 83):

- **Longitudinal plane crossing column center**
- **Transverse plane crossing column center**

Distances away from the column center are calculated to match the corresponding soil nodes and are listed in the Middle Dropdown List (Fig. 84). Fig. 85 is the sample output of the soil settlement time histories under the left abutment.

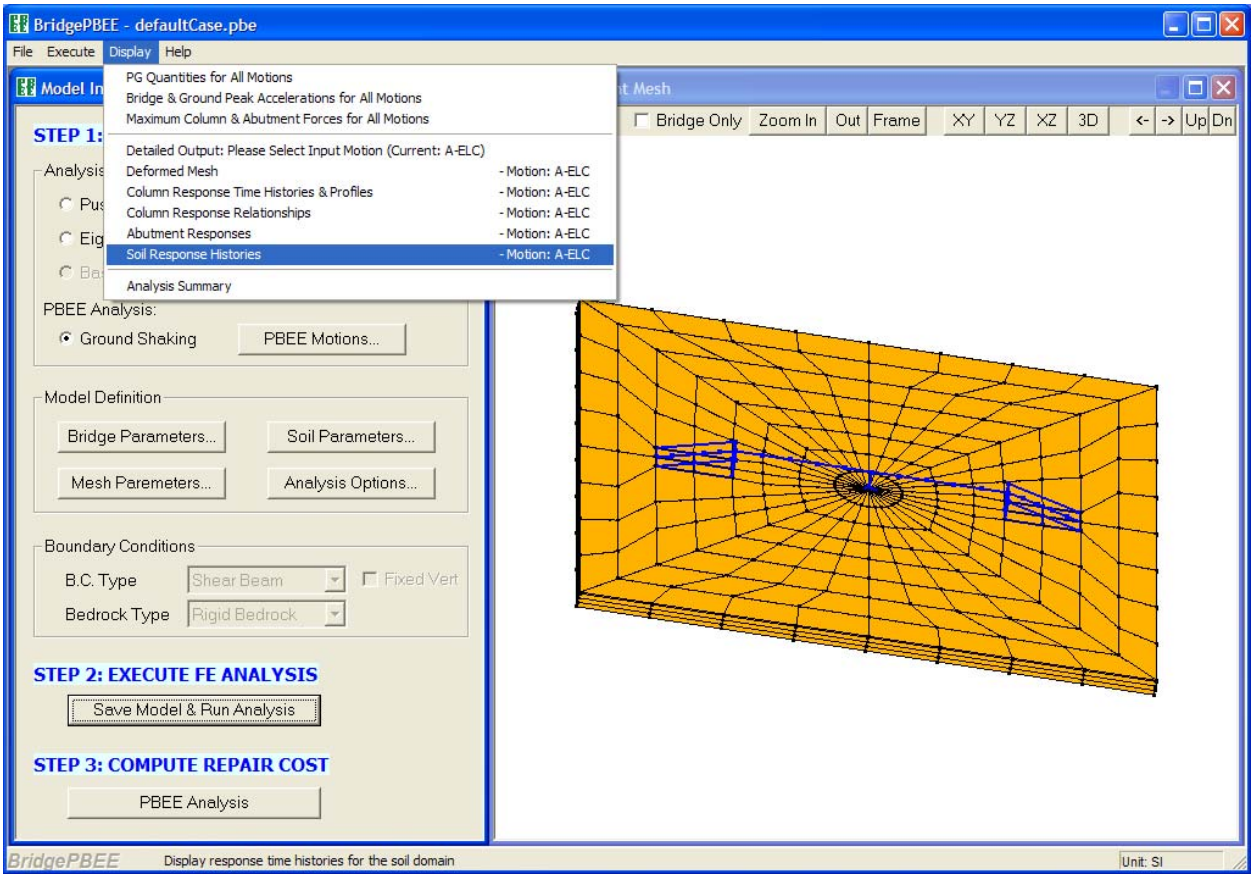


Fig. 81. Menu items to access the soil responses

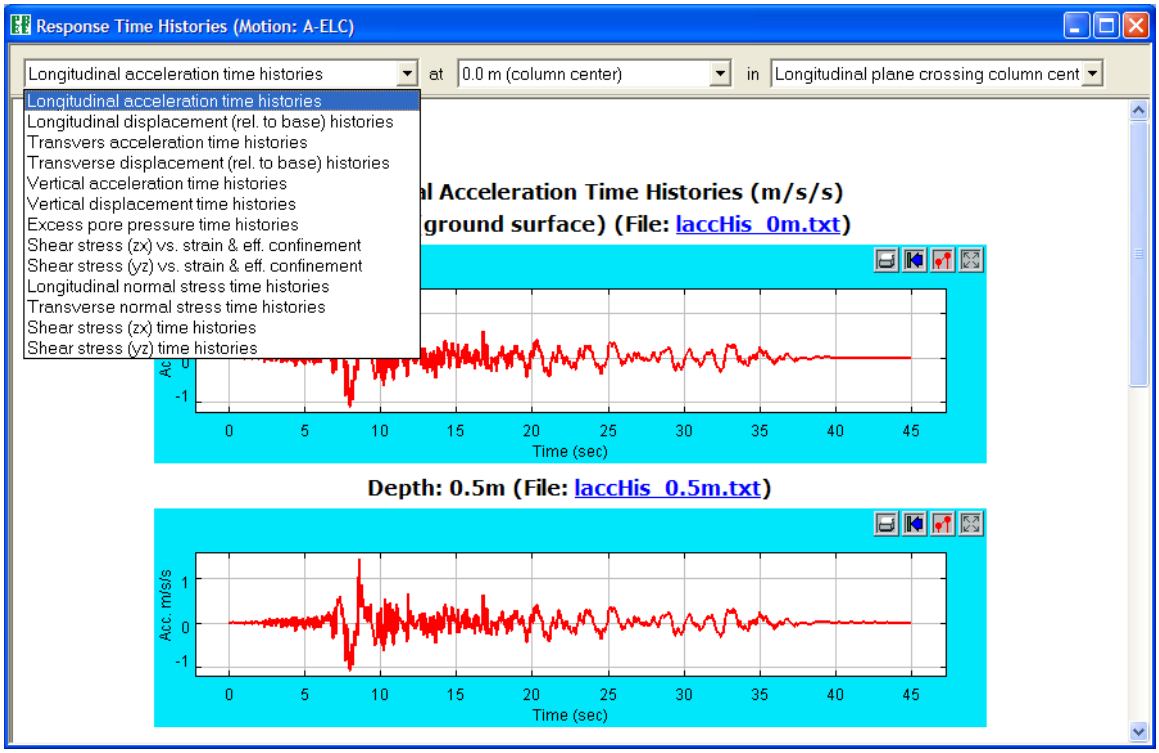

Response Time Histories (Motion: A-ELC)
Longitudinal acceleration time histories
at
0.0 m (column center)
in
Longitudinal plane crossing column cent
Longitudinal acceleration time histories
Longitudinal displacement (rel. to base) histories
Transvers acceleration time histories
Transverse displacement (rel. to base) histories
Vertical acceleration time histories
Vertical displacement time histories
Excess pore pressure time histories
Shear stress (zx) vs. strain & eff. confinement
Shear stress (yz) vs. strain & eff. confinement
Longitudinal normal stress time histories
Transverse normal stress time histories
Shear stress (zx) time histories
Shear stress (yz) time histories
l Acceleration Time Histories (m/s/s)
ground surface) (File: laccHis_0m.txt)
Time (sec)
Depth: 0.5m (File: laccHis_0.5m.txt)
Acc. m/s/s
Time (sec)

Fig. 82. Response options for soil time histories

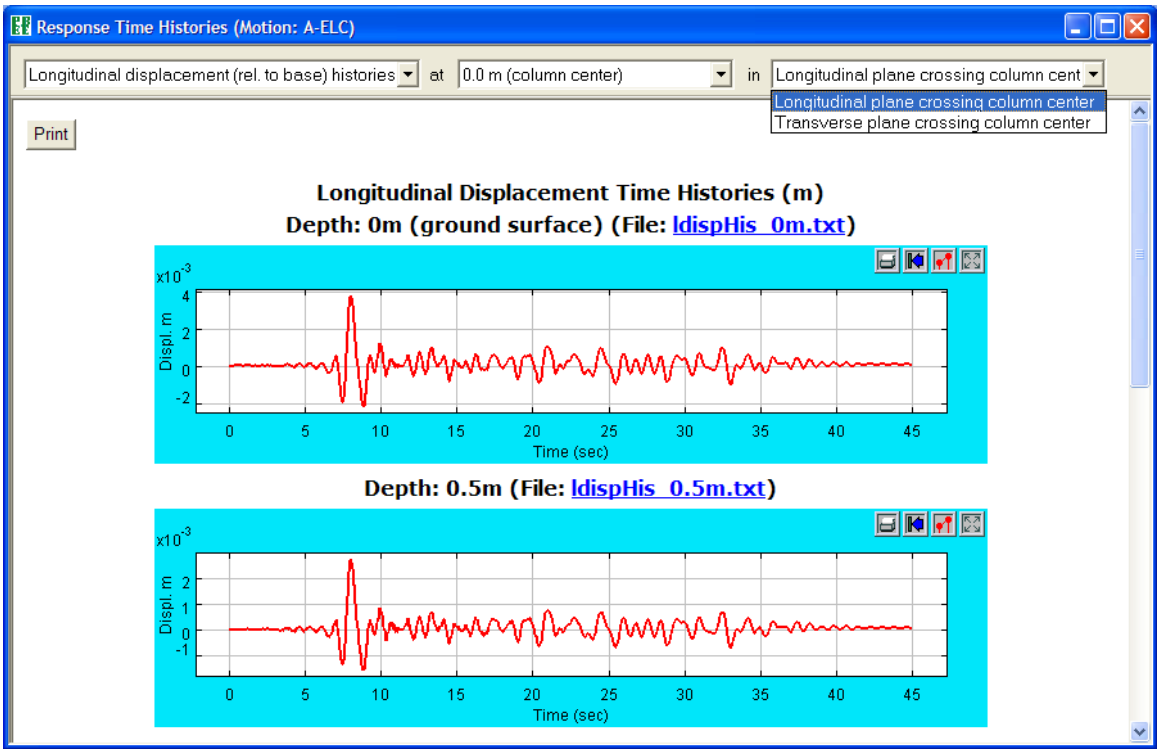


Fig. 83. Planes for locations of the soil response time histories

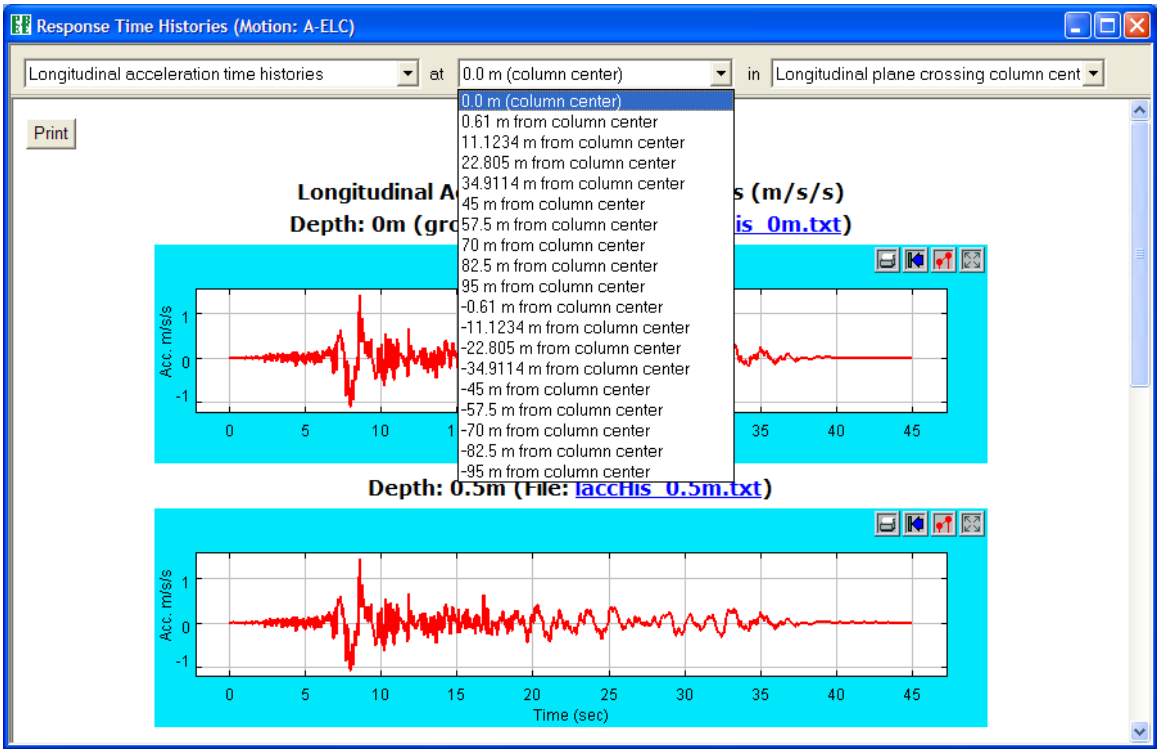

Response Time Histories (Motion: A-ELC)
Longitudinal acceleration time histories
at
0.0 m (column center)
in
Longitudinal plane crossing column cent
Print
0.0 m (column center)
0.61 m from column center
11.1234 m from column center
22.805 m from column center
34.9114 m from column center
45 m from column center
57.5 m from column center
70 m from column center
82.5 m from column center
95 m from column center
-0.61 m from column center
-11.1234 m from column center
-22.805 m from column center
-34.9114 m from column center
-45 m from column center
-57.5 m from column center
-70 m from column center
-82.5 m from column center
-95 m from column center
Depth: 0m (gr
is_0m.txt)
Depth: 0.5m (File: TaccHis_0.5m.txt)
Acc. m/s/s
Time (sec)

Fig. 84. Locations of soil response time histories

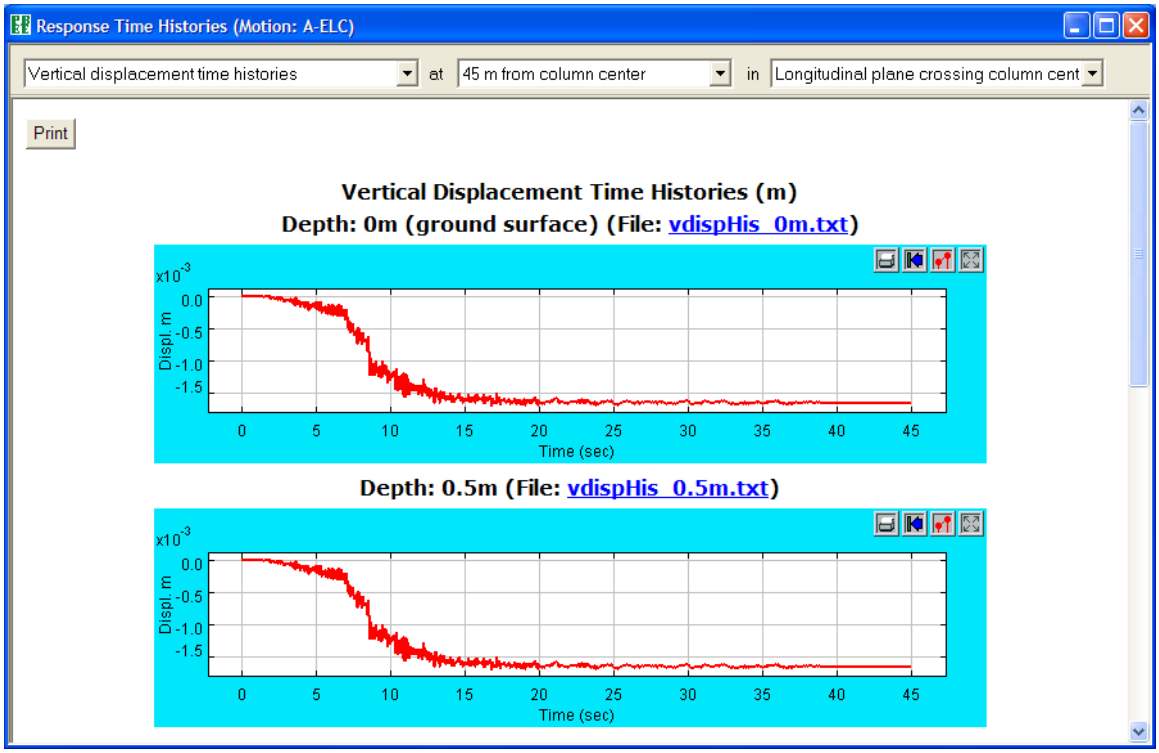


Fig. 85. Soil settlement time histories under abutment

7.1.6 PBEE Output Quantities

At the end of the finite element analysis phase, the following output performance group quantities (for each earthquake record) are used in the next phase of PBEE analysis:

Table 11. PBEE Performance Groups

Performance Group (PG) #	Performance group names
1	Maximum column drift ratio
2	Residual column drift ratio
3	Maximum relative deck-end/abutment displacement (left)
4	Maximum relative deck-end/abutment displacement (right)
5	Maximum bridge-abutment bearing displacement (left)
6	Maximum bridge-abutment bearing displacement (right)
7	Approach residual vertical displacement (left)
8	Approach residual vertical displacement (right)
9	Abutment residual pile cap displacement (left)
10	Abutment residual pile top displacement (right)
11	Column residual pile displacement at ground surface

In addition, Intensity Measures for the computed Free Field ground surface acceleration records are computed, so that outcomes can be either shown against the input base shaking IMs or the computed ground surface IMs (noted as Free-Field in the user interface). The sections below detail how the response quantities are obtained for each PG. Refer to Fig. 86 for the annotated model that is used to describe the location of sampling points during time history analysis.

PG1: Maximum tangential drift ratio SRSS (column)
PG2: Residual tangential drift ratio SRSS (column)

The tangential drift ratio is defined as the maximum of a) displacement above the inflection point divided by the length of this distance, and b) displacement below the inflection point divided by the length of this distance. This takes care of rotation at the base, different boundary conditions, etc., so that the results are consistent when computing damage. The Square Root of Sum of Squares (SRSS) values of the 2 horizontal components are used. The tangential drift ratios are combined separately at each time step (to obtain SRSS).

PG1 (Max tangential drift ratio SRSS) is the maximum of the SRSS values of all time steps. PG2 (Residual tangential drift ratio SRSS) is the SRSS value at the last time step. The tangential drift ratio is in percentage.

To calculate the tangential drift ratio, the following 2 lines were added into the tcl file:

```
recorder Element -file A-ELC.dft -time -ele $columnEle tangentDrift
recorder Element -file A-ELC.ifp -time -ele $columnEle inflectionPoint
```

where `$columnEle` is the element # of the column (Only one forced-based beam-column element `nonlinearBeamColumn` is used for the column). In the `.dft` file, there will be 5 columns of data for each time step and the first column is time. In the `.ifp` file, there will be 3 columns for each time step and the first column is also time.

Subsequently, the tangential drift ratio is calculated using the code snippet shown in Fig. 87. For the tangential drift ratio in the longitudinal direction (X-direction or bridge longitudinal direction), the `tdx1` and `tdx2` variables are the second and third column (the first column is time), respectively, of the tangential drift recorder file (e.g., `A-ELC.dft`). The `tdxi` variable is the second column (the first column is time) of the inflection point recorder file (e.g., `A-ELC.ifp`).

For the transverse tangential drift ratio, the `tdx1` and `tdx2` variables are the fourth and fifth column of the `.dft` file and the `tdxi` variable is the third column of the `.ifp` file.

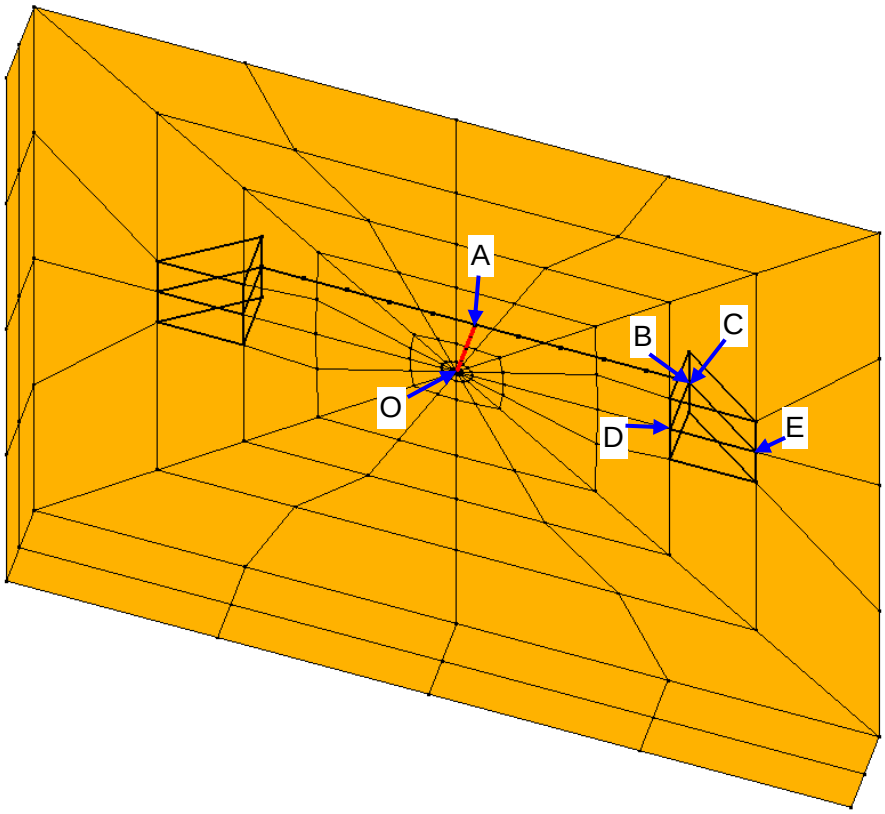


Fig. 86. Finite element mesh in BridgePBEE: Node O – Column base node (at ground surface); Node A – Column top node; Node B – Deck-end node; Node C – Abutment top node (having the same coordinates as Node B); Nodes B are C are connected by an abutment model; Node D – Abutment pile cap node

```
// tdx1 & tdx2 -- the tangent drift recorder file at time step i
// tdxi -- the inflection point recorder at time step i
// tdx – tangential drift ratio

if( fabs(tdxi) < 1e-20 ) {
    tdx = -tdx2/(H - tdxi);
}
else if ( fabs(H-tdxi) < 1e-20 ) {
    tdx = -tdx1 / tdxi;
}
else {
    tdx = __max(fabs(tdx1/tdxi), fabs(tdx2/(H-tdxi)));
    //tdx = -tdx*sgn(tdx2/(H-tdxi));
    if( fabs(tdx2/(H-tdxi)) < 1e-20 )
        tdx = 0;
    else if( (tdx2/(H-tdxi)) > 0)
        tdx = -tdx;
}
return tdx;
```

Fig. 87. Code snippet to calculate the tangential drift ratio of column

PG3: Maximum longitudinal relative deck-end/abutment displacement (left)
PG4: Maximum longitudinal relative deck-end/abutment displacement (right)

These two PGs are intended to address the issue of abutment impact into the backwall, so they are defined as only the motion of the deck into the abutment. Maximum absolute values in the longitudinal direction are used.

For example, for the right abutment shown in Fig. 86, it is the relative longitudinal displacement of node B (deck-end node) in the direction of node C (abutment top node). A zero value is used for the times during which the deck-end node moves away from the abutment top node.

PG5: Maximum absolute bearing displacement (left abutment)
PG6: Maximum absolute bearing displacement (right abutment)

These two PGs are intended to address bearing damage whether or not an explicit representation of the bearings is included in the user-selected abutment model. Therefore, the EDP for the PG is based on the relative displacements of the deck-end node (e.g., Node B for the right abutment shown in Fig. 86) to the abutment top node (e.g., Node C for right abutment shown in Fig. 86). The SRSS values of the resulting two relative horizontal displacements is used and both motion into the backwall and away from the backwall are considered.

PG7: Residual vertical displacement (left abutment)
PG8: Residual vertical displacement (right abutment)

This PG is used to gage immediate repairs for rideability, and is not a measure of the permanent slumping of the embankment (for example). Therefore, the EDP is calculated as the vertical displacement of the abutment top node (e.g., Node B for the right abutment shown in Fig. 86) relative to the deck-end node (e.g., Node C for the right abutment shown in Fig. 86). The residual value is used (value at the final time step).

PG9: Residual pile cap displacement SRSS (left abutment)
PG10: Residual pile cap displacement SRSS (right abutment)

These PGs address possible damage below grade due to lateral translation of the piles and pile caps. While not a direct measure, pile cap displacement was selected as it would not require knowledge or observations of piles below grade. The EDP is defined by calculating the SRSS value of the 2 horizontal displacements of the abutment pile cap node (e.g., Node D for the right abutment shown in Fig. 86). The residual is obtained from the value at the final time step.

PG11: Residual pile cap displacement SRSS (column)

This quantity is analogous to the two previous PG, but is representative of response and damage at the abutment foundations. The EDP is obtained by calculating the SRSS value of the 2 horizontal displacements of the column pile cap node (e.g., Node O shown in Fig. 86) and taking the value at the final time step.

The PG (Performance Group) quantities for all input motions can be accessed by clicking menu **Display** (Fig. 3) and then **PG Quantities for All Motion** (Fig. 88). The window to display PG quantities is shown in Fig. 89.

The PG quantities are displayed against any of the 22 intensity measures (including 11 for the input acceleration and the other 11 for the free-field response). The PG quantities for each input motion are displayed by bin of the motion (see legend in Fig. 89). When an IM is paired with an EDP and all the individual realizations are plotted, the result is typically termed a demand model, or probabilistic seismic demand model (PSDM). Previous research has demonstrated that the central values of PSDMs are often well described using a power-law relationship between EDP and IM. The parameters of such a power-law fit can be obtained using least squares analysis on the data. Therefore, when plotted in log-log space (as is shown in Fig. 89), the best-fit, or mean, relationship is linear.

The mean (in log-log space) is shown along with the standard deviation (also in log-log space) of the power-law fit. If it is assumed that the EDP responses are lognormally distributed when conditioned on IM, then these curves can be interpreted as being defined by the two parameters of a lognormal distribution (the median can be related to the mean of the logarithm of the data and the lognormal standard deviation is as shown).

To convert all figures currently displayed in the window, click **Convert Figures to EPS Format**. A MS Word window with the EPS figures included in the document will pop up once the converting is done (Fig. 90).

To view lognormal standard deviations for each PG (Fig. 91), click **View Beta Values** in Fig. 89. The information is tabulated with values in bold indicating the lowest lognormal standard deviation for all the computed IMs in a given PG. The same information is shown graphically as a bar chart (separate bar chart for each PG). Such a figure is useful for determining the selection of optimal IM for a given EDP or PG.

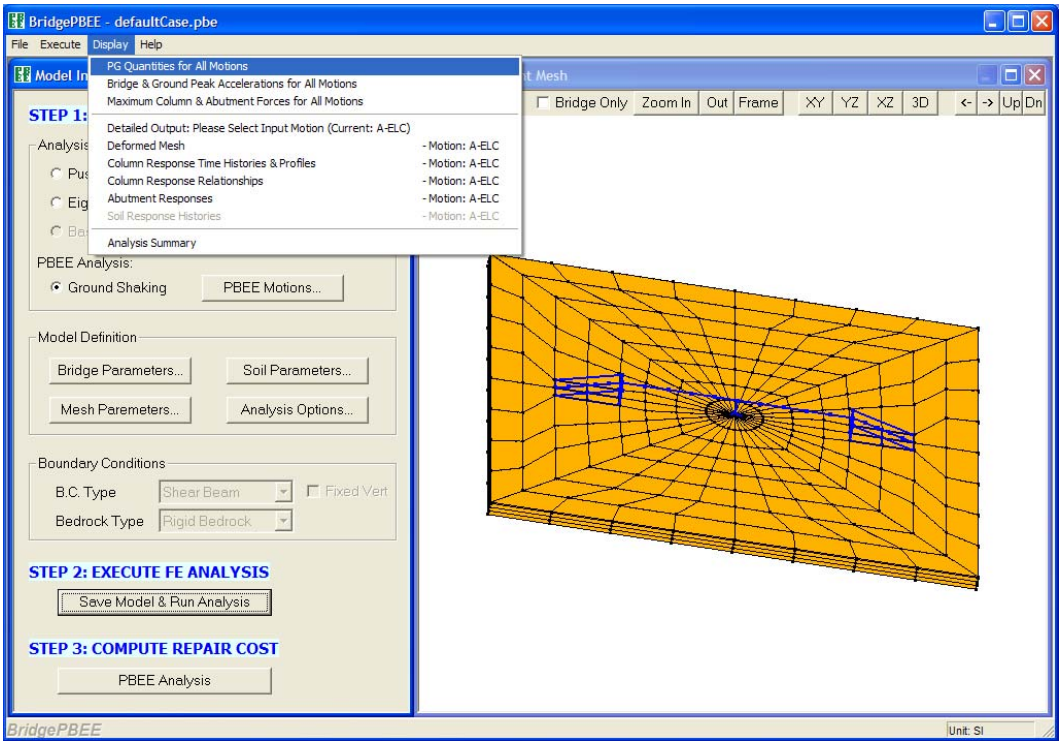


Fig. 88. Menu items to access the PG quantities for all motions

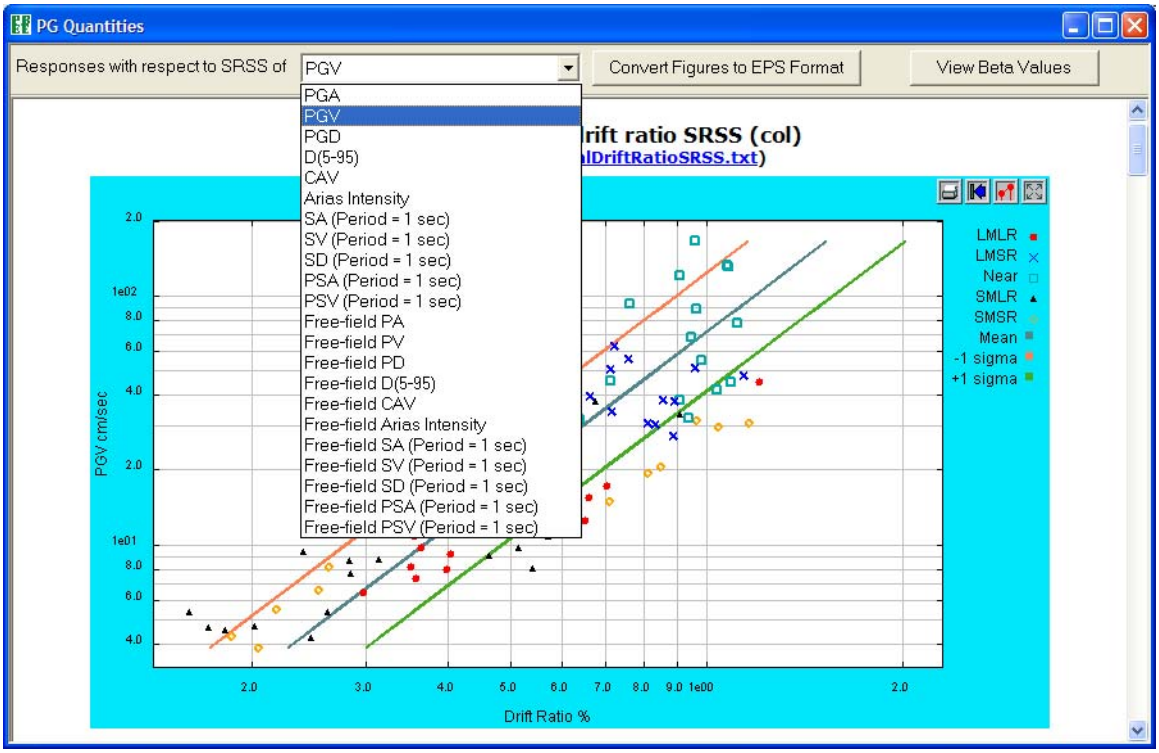


Fig. 89. PG quantities for all motions (scroll down to see all 11 PGs)

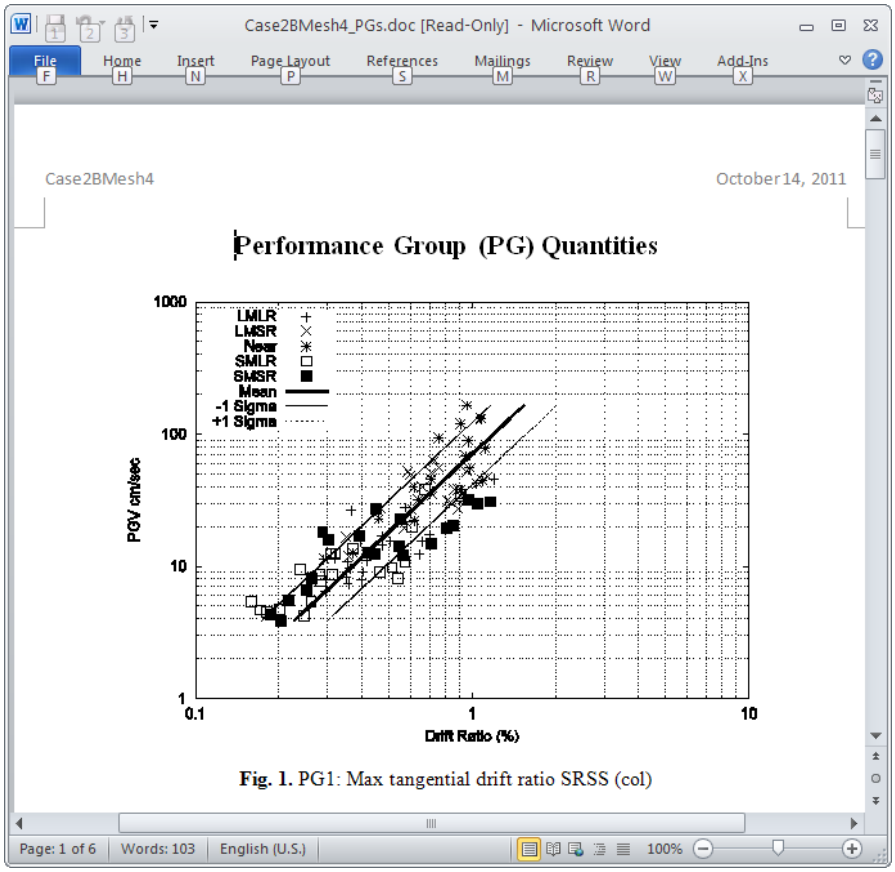
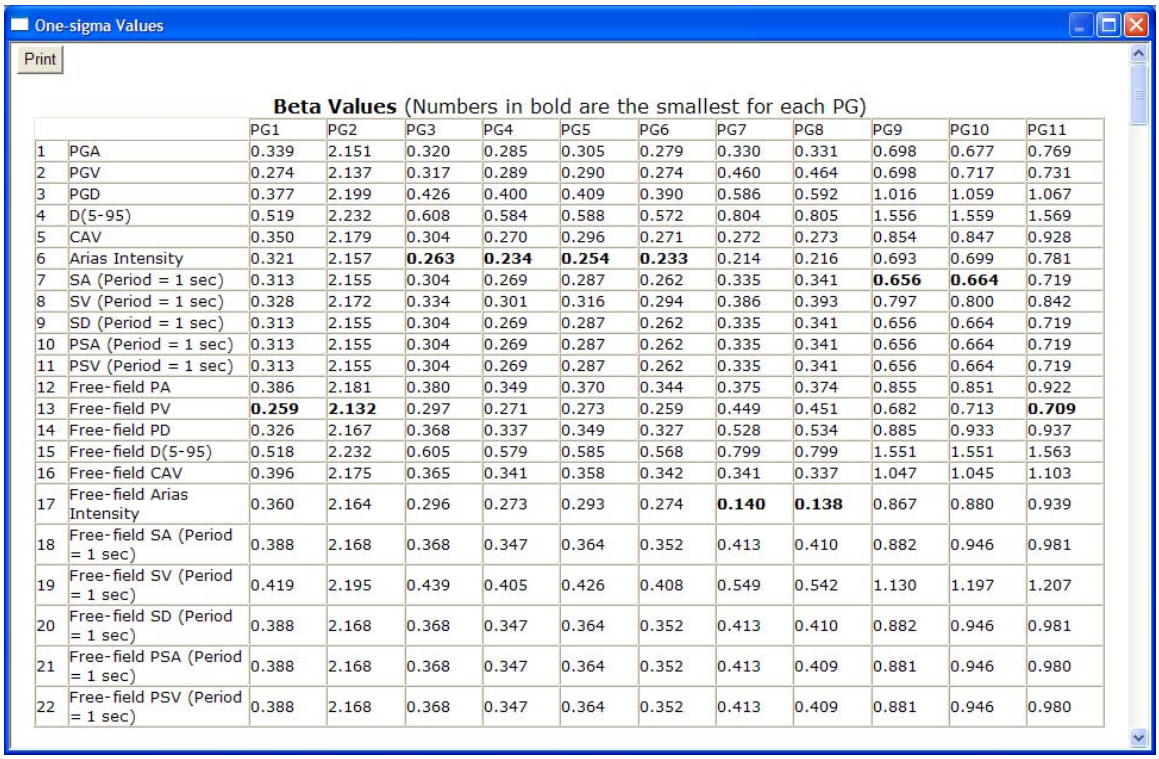


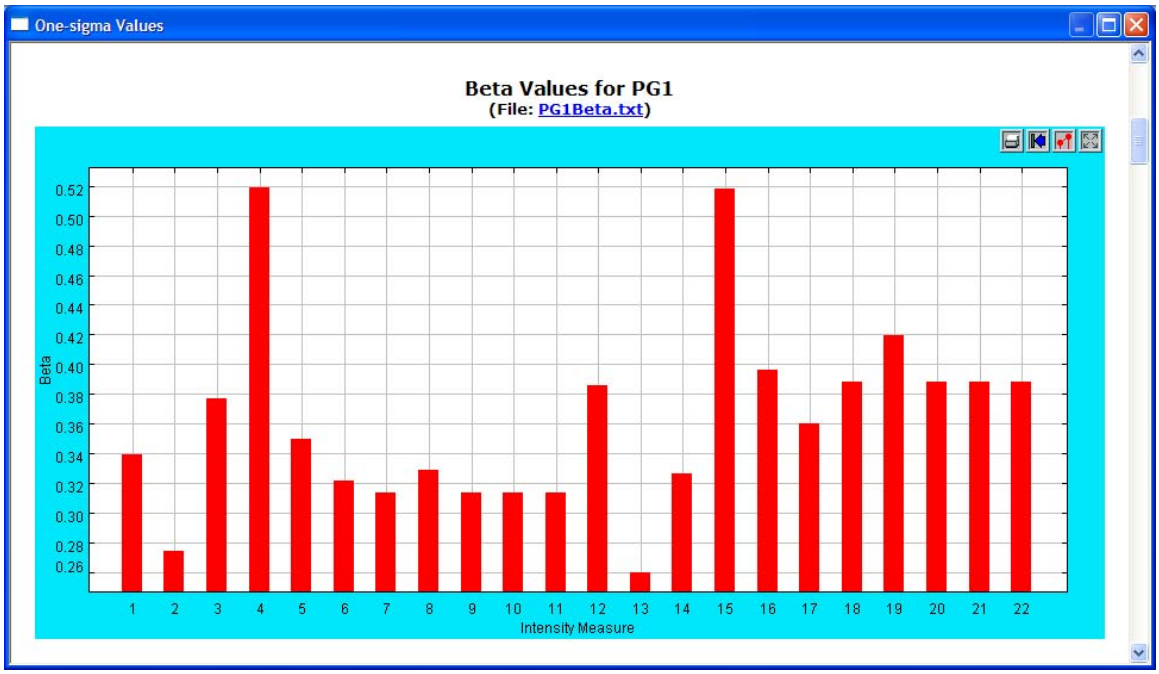
Fig. 90. Converting figures to EPS format



Beta Values (Numbers in bold are the smallest for each PG)

		PG1	PG2	PG3	PG4	PG5	PG6	PG7	PG8	PG9	PG10	PG11
1	PGA	0.339	2.151	0.320	0.285	0.305	0.279	0.330	0.331	0.698	0.677	0.769
2	PGV	0.274	2.137	0.317	0.289	0.290	0.274	0.460	0.464	0.698	0.717	0.731
3	PGD	0.377	2.199	0.426	0.400	0.409	0.390	0.586	0.592	1.016	1.059	1.067
4	D(5-95)	0.519	2.232	0.608	0.584	0.588	0.572	0.804	0.805	1.556	1.559	1.569
5	CAV	0.350	2.179	0.304	0.270	0.296	0.271	0.272	0.273	0.854	0.847	0.928
6	Arias Intensity	0.321	2.157	**0.263**	**0.234**	**0.254**	**0.233**	0.214	0.216	0.693	0.699	0.781
7	SA (Period = 1 sec)	0.313	2.155	0.304	0.269	0.287	0.262	0.335	0.341	**0.656**	**0.664**	0.719
8	SV (Period = 1 sec)	0.328	2.172	0.334	0.301	0.316	0.294	0.386	0.393	0.797	0.800	0.842
9	SD (Period = 1 sec)	0.313	2.155	0.304	0.269	0.287	0.262	0.335	0.341	0.656	0.664	0.719
10	PSA (Period = 1 sec)	0.313	2.155	0.304	0.269	0.287	0.262	0.335	0.341	0.656	0.664	0.719
11	PSV (Period = 1 sec)	0.313	2.155	0.304	0.269	0.287	0.262	0.335	0.341	0.656	0.664	0.719
12	Free-field PA	0.386	2.181	0.380	0.349	0.370	0.344	0.375	0.374	0.855	0.851	0.922
13	Free-field PV	**0.259**	**2.132**	0.297	0.271	0.273	0.259	0.449	0.451	0.682	0.713	**0.709**
14	Free-field PD	0.326	2.167	0.368	0.337	0.349	0.327	0.528	0.534	0.885	0.933	0.937
15	Free-field D(5-95)	0.518	2.232	0.605	0.579	0.585	0.568	0.799	0.799	1.551	1.551	1.563
16	Free-field CAV	0.396	2.175	0.365	0.341	0.358	0.342	0.341	0.337	1.047	1.045	1.103
17	Free-field Arias Intensity	0.360	2.164	0.296	0.273	0.293	0.274	**0.140**	**0.138**	0.867	0.880	0.939
18	Free-field SA (Period = 1 sec)	0.388	2.168	0.368	0.347	0.364	0.352	0.413	0.410	0.882	0.946	0.981
19	Free-field SV (Period = 1 sec)	0.419	2.195	0.439	0.405	0.426	0.408	0.549	0.542	1.130	1.197	1.207
20	Free-field SD (Period = 1 sec)	0.388	2.168	0.368	0.347	0.364	0.352	0.413	0.410	0.882	0.946	0.981
21	Free-field PSA (Period = 1 sec)	0.388	2.168	0.368	0.347	0.364	0.352	0.413	0.409	0.881	0.946	0.980
22	Free-field PSV (Period = 1 sec)	0.388	2.168	0.368	0.347	0.364	0.352	0.413	0.409	0.881	0.946	0.980

a)



b)

Fig. 91. Lognormal standard deviations (beta values) for each PG: a) table format; b) bar graph format

7.1.7 Bridge Peak Accelerations for All Motions

The bridge peak accelerations for all input motions can be accessed by clicking menu **Display** (Fig. 3) and then **Bridge Peak Accelerations for All Motions** (Fig. 92). The window to display the bridge peak accelerations for all motions is shown in Fig. 93. The responses are available in the longitudinal and transverse directions as well as for the SRSS of the 2 horizontal directions (Fig. 93).

The figures in this window include (The free-field location is defined in Fig. 66):

- Maximum bridge acceleration
- Maximum column base acceleration
- Maximum free-field acceleration
- Maximum input acceleration
- Bridge peak acceleration / column base peak acceleration
- Column base peak acceleration / input peak acceleration
- Free-field peak acceleration / input peak acceleration
- Bridge peak acceleration / input peak acceleration

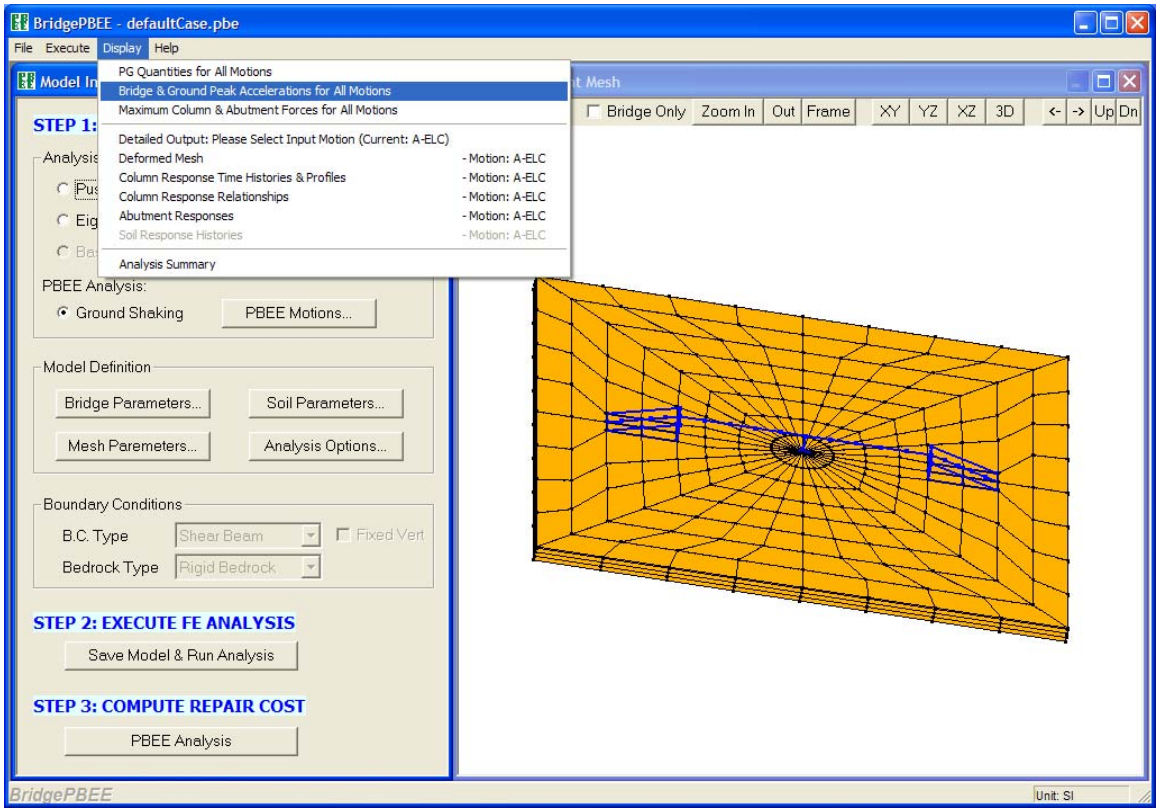
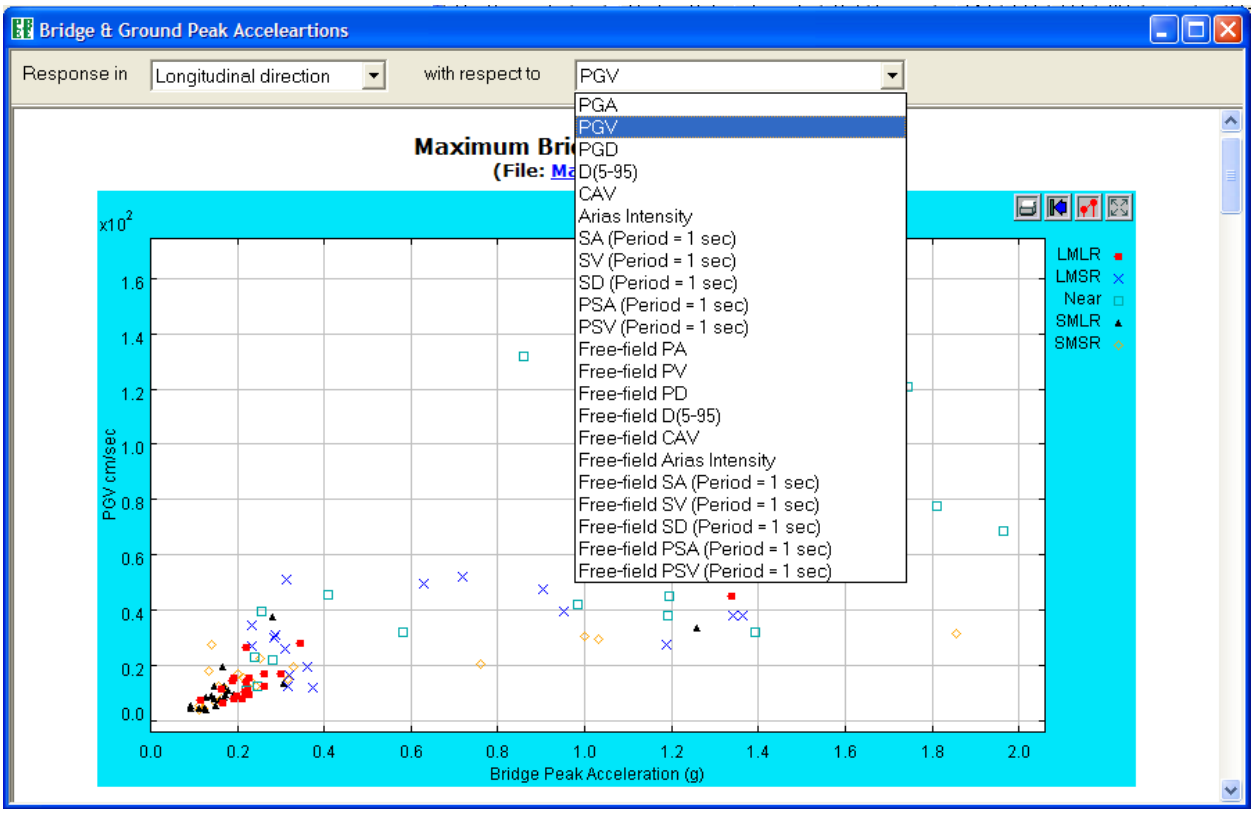
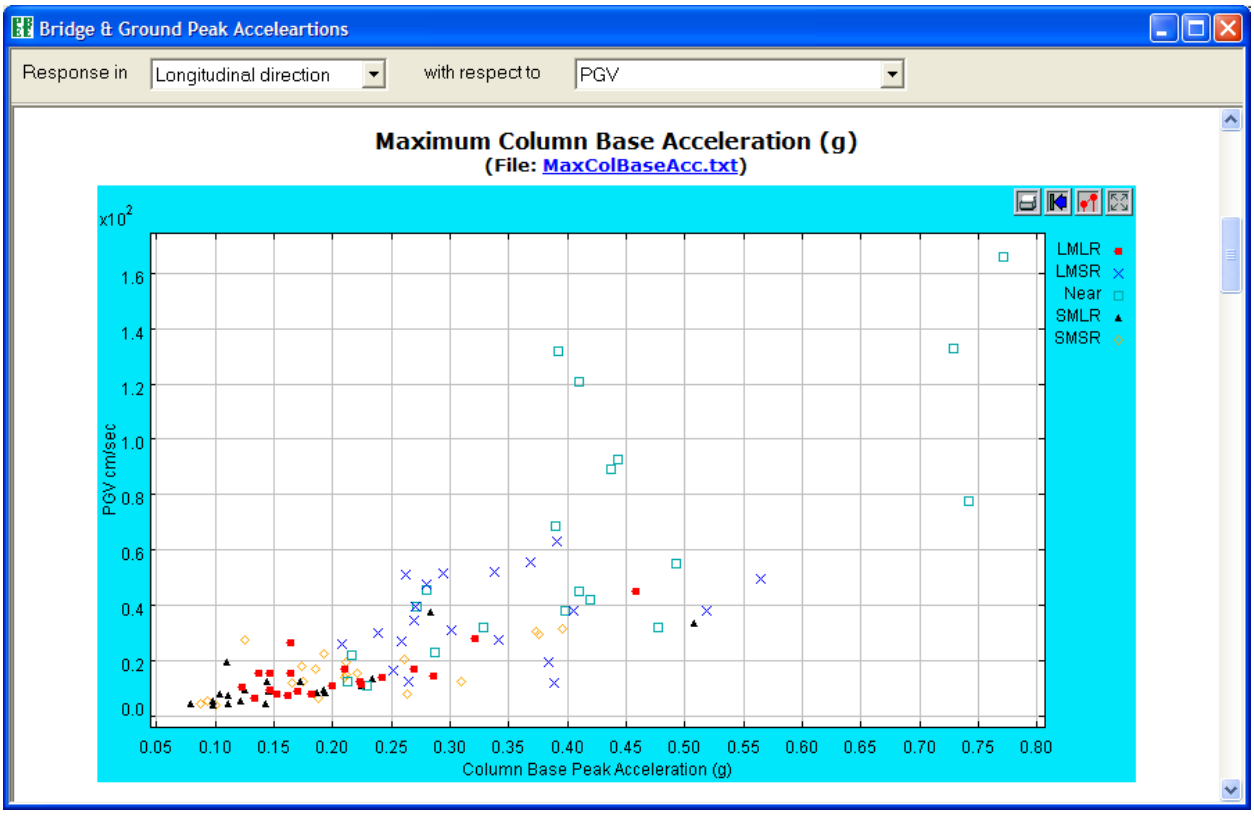


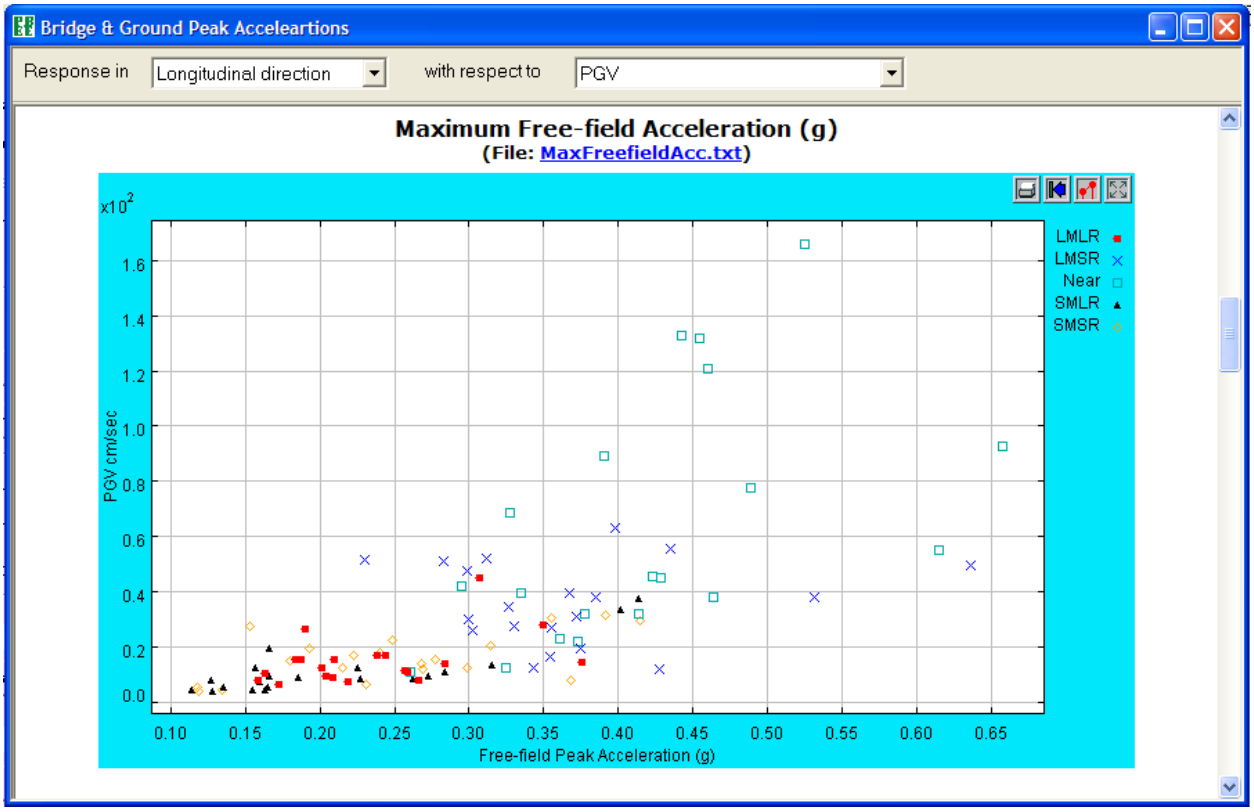
Fig. 92. Menu items to access bridge peak accelerations for all motions


Bridge & Ground Peak Acceleartions
Response in
Longitudinal direction
with respect to
PGV
PGA
PGV
PGD
D(5-95)
CAV
Arias Intensity
SA (Period = 1 sec)
SV (Period = 1 sec)
SD (Period = 1 sec)
PSA (Period = 1 sec)
PSV (Period = 1 sec)
Free-field PA
Free-field PV
Free-field PD
Free-field D(5-95)
Free-field CAV
Free-field Arias Intensity
Free-field SA (Period = 1 sec)
Free-field SV (Period = 1 sec)
Free-field SD (Period = 1 sec)
Free-field PSA (Period = 1 sec)
Free-field PSV (Period = 1 sec)
Maximum Bri
(File: Ma
x10^2
PGV cm/sec
Bridge Peak Acceleration (g)
LMLR
LMSR
Near
SMLR
SMSR

a)


Bridge & Ground Peak Acceleartions
Response in
Longitudinal direction
with respect to
PGV
Maximum Column Base Acceleration (g)
(File: MaxColBaseAcc.txt)
x10^2
PGV cm/sec
Column Base Peak Acceleration (g)
LMLR
LMSR
Near
SMLR
SMSR

b)



c)

Fig. 93. Bridge peak accelerations for all motions: a) maximum bridge accelerations; b) maximum column base accelerations; and c) maximum free-field accelerations

7.1.8 Maximum Column & Abutment Forces for All Motions

The maximum column & abutment forces for all input motions can be accessed by clicking menu **Display** (Fig. 3) and then **Maximum Column & Abutment Forces for All Motions** (Fig. 94). The window to display the maximum column & abutment forces for all motions is shown in Fig. 95. The responses are available in the longitudinal and transverse directions as well as for the SRSS of the 2 horizontal directions (Fig. 95).

The figures in this window include:

- Maximum column shear forces
- Maximum column bending moments
- Maximum abutment forces (left abutment)
- Maximum abutment forces (right abutment)

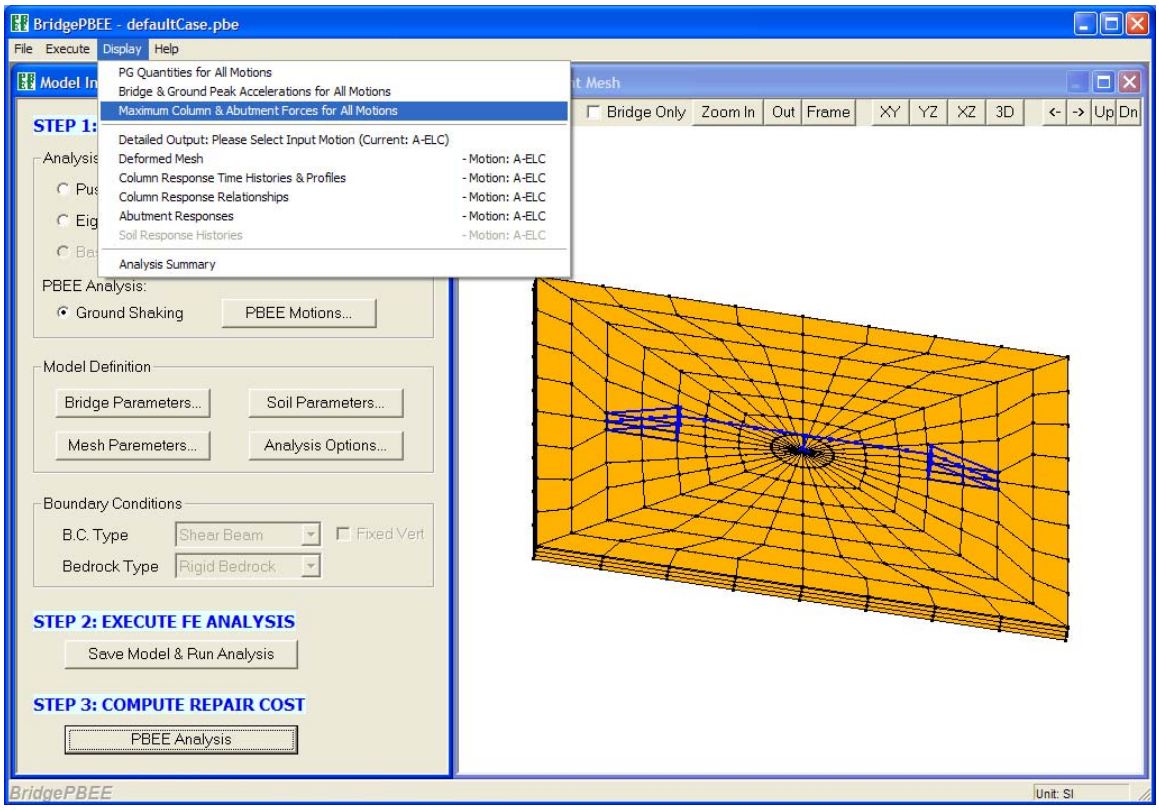
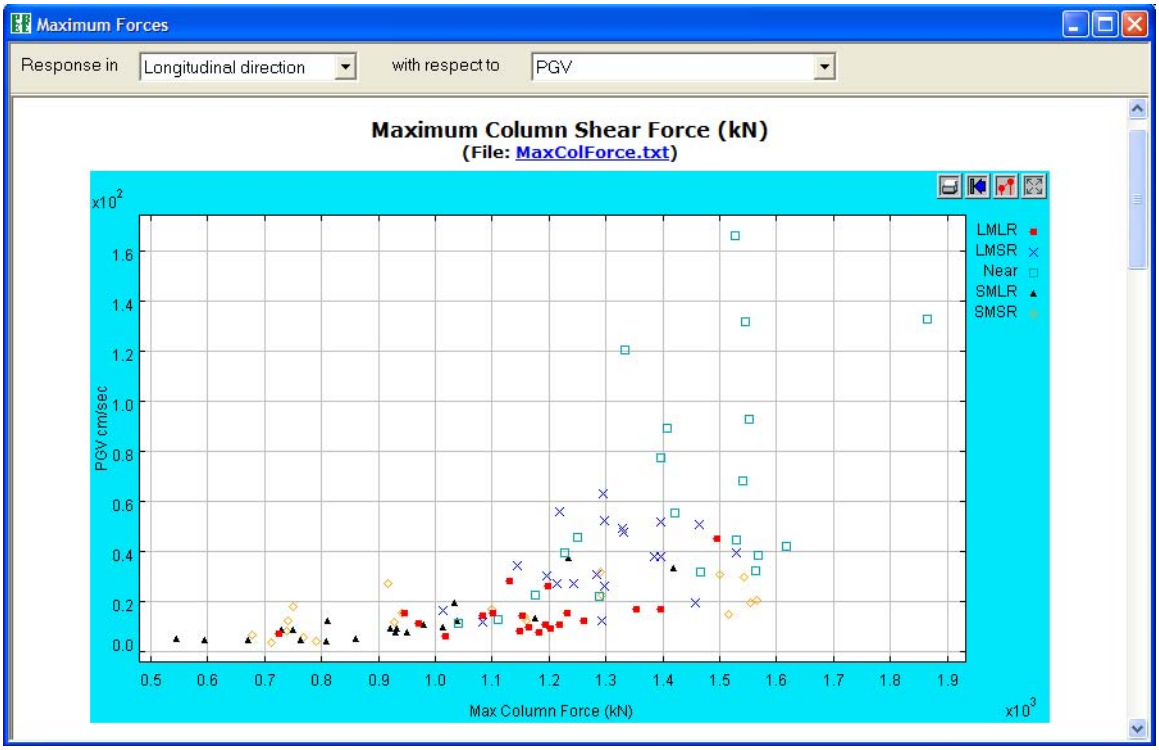
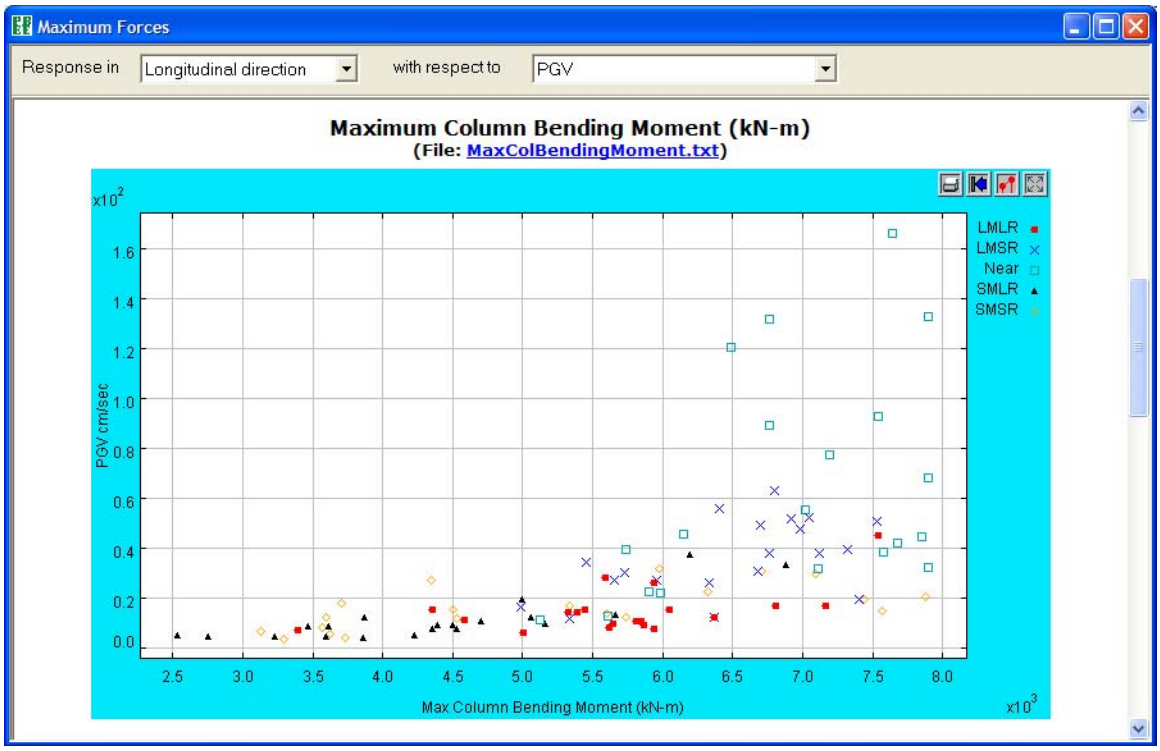


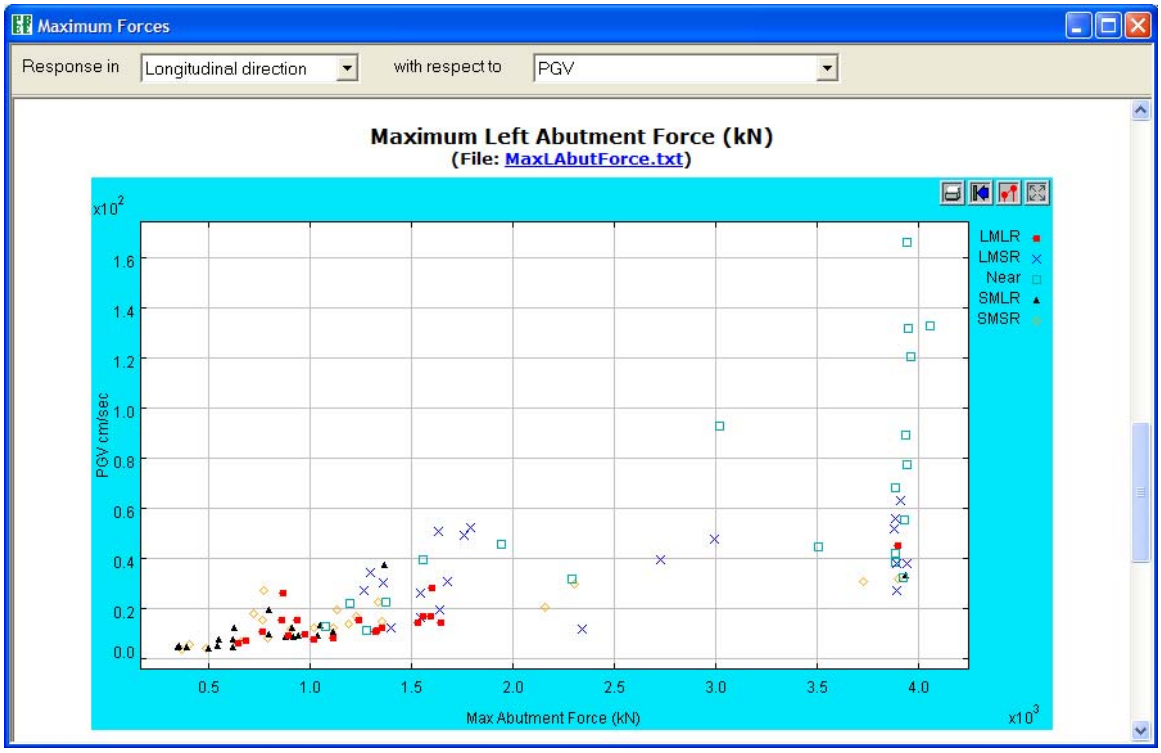
Fig. 94. Menu items to access maximum column & abutment forces for all motions



a)



b)



c)

Fig. 95. Maximum column & abutment forces for all motions: a) maximum column shear forces; b) maximum column bending moments; and c) maximum abutment forces

7.2 PBEE Outcomes

7.2.1 Repair Cost & Time

The final PBEE results will be displayed against any intensity measure (e.g., PGV) in terms of:

- Contribution to expected repair cost ($) from each performance group (Fig. 96)
- Total repair cost ratio (%) (Fig. 97)
- Contribution to expected repair cost ($) from each repair quantity (Fig. 98)
- Contribution to repair cost standard deviation ($) from each repair quantity (Fig. 99)
- Total repair time (CWD) where CWD stands for Crew Working Day (Fig. 100)
- Contribution to expected repair time (CWD) from each repair quantity (Fig. 101)

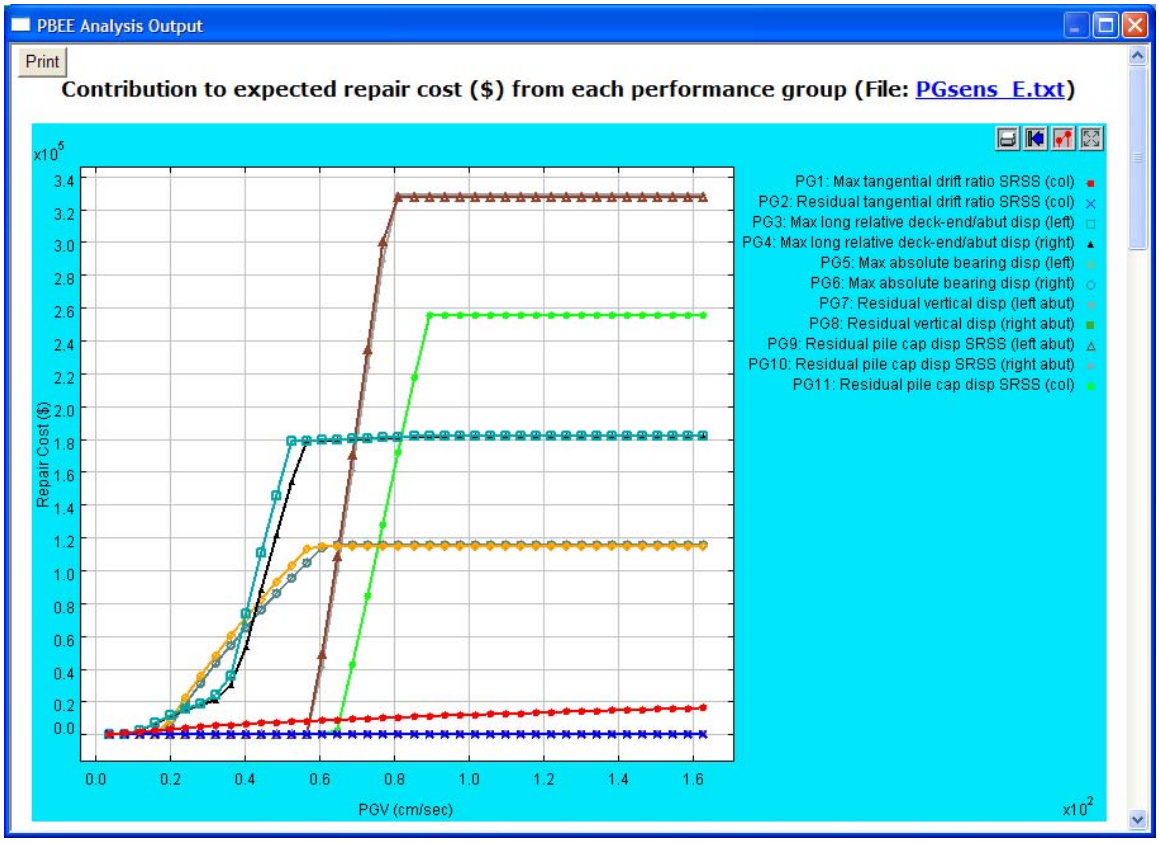

PBEE Analysis Output
Print
Contribution to expected repair cost ($) from each performance group (File: PGsens_E.txt)
x10^5
Repair Cost ($)
PGV (cm/sec)
x10^2
PG1: Max tangential drift ratio SRSS (col)
PG2: Residual tangential drift ratio SRSS (col)
PG3: Max long relative deck-end/abut disp (left)
PG4: Max long relative deck-end/abut disp (right)
PG5: Max absolute bearing disp (left)
PG6: Max absolute bearing disp (right)
PG7: Residual vertical disp (left abut)
PG8: Residual vertical disp (right abut)
PG9: Residual pile cap disp SRSS (left abut)
PG10: Residual pile cap disp SRSS (right abut)
PG11: Residual pile cap disp SRSS (col)

Fig. 96. Contribution to expected repair cost ($) from each performance group

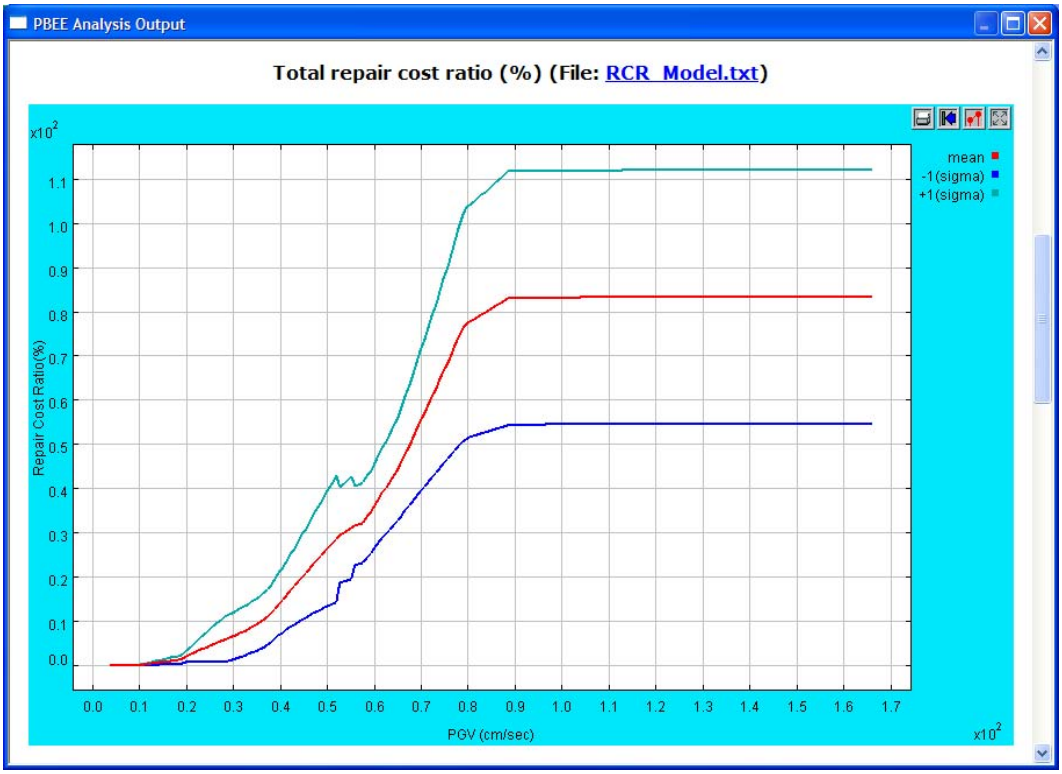


Fig. 97. Total repair cost ratio (%) as a function of intensity

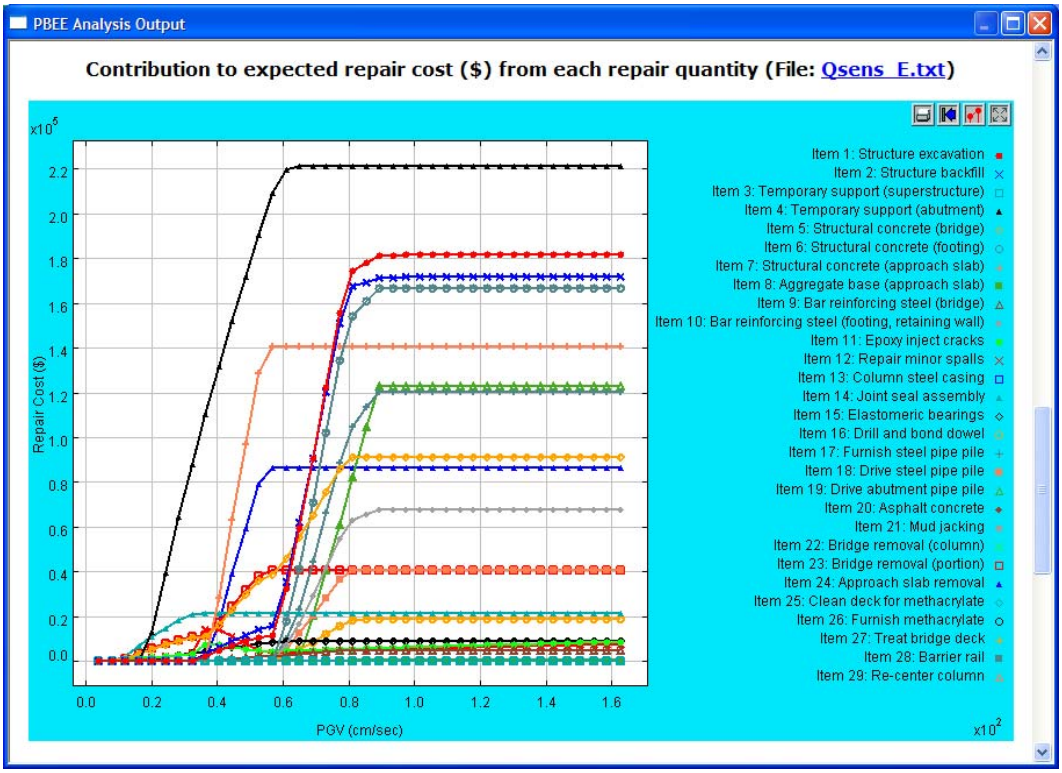

PBEE Analysis Output
Contribution to expected repair cost ($) from each repair quantity (File: Qsens_E.txt)
x10^5
Repair Cost ($)
PGV (cm/sec)
x10^2
Item 1: Structure excavation
Item 2: Structure backfill
Item 3: Temporary support (superstructure)
Item 4: Temporary support (abutment)
Item 5: Structural concrete (bridge)
Item 6: Structural concrete (footing)
Item 7: Structural concrete (approach slab)
Item 8: Aggregate base (approach slab)
Item 9: Bar reinforcing steel (bridge)
Item 10: Bar reinforcing steel (footing, retaining wall)
Item 11: Epoxy inject cracks
Item 12: Repair minor spalls
Item 13: Column steel casing
Item 14: Joint seal assembly
Item 15: Elastomeric bearings
Item 16: Drill and bond dowel
Item 17: Furnish steel pipe pile
Item 18: Drive steel pipe pile
Item 19: Drive abutment pipe pile
Item 20: Asphalt concrete
Item 21: Mud jacking
Item 22: Bridge removal (column)
Item 23: Bridge removal (portion)
Item 24: Approach slab removal
Item 25: Clean deck for methacrylate
Item 26: Furnish methacrylate
Item 27: Treat bridge deck
Item 28: Barrier rail
Item 29: Re-center column

Fig. 98. Contribution to expected repair cost ($) from each repair quantity

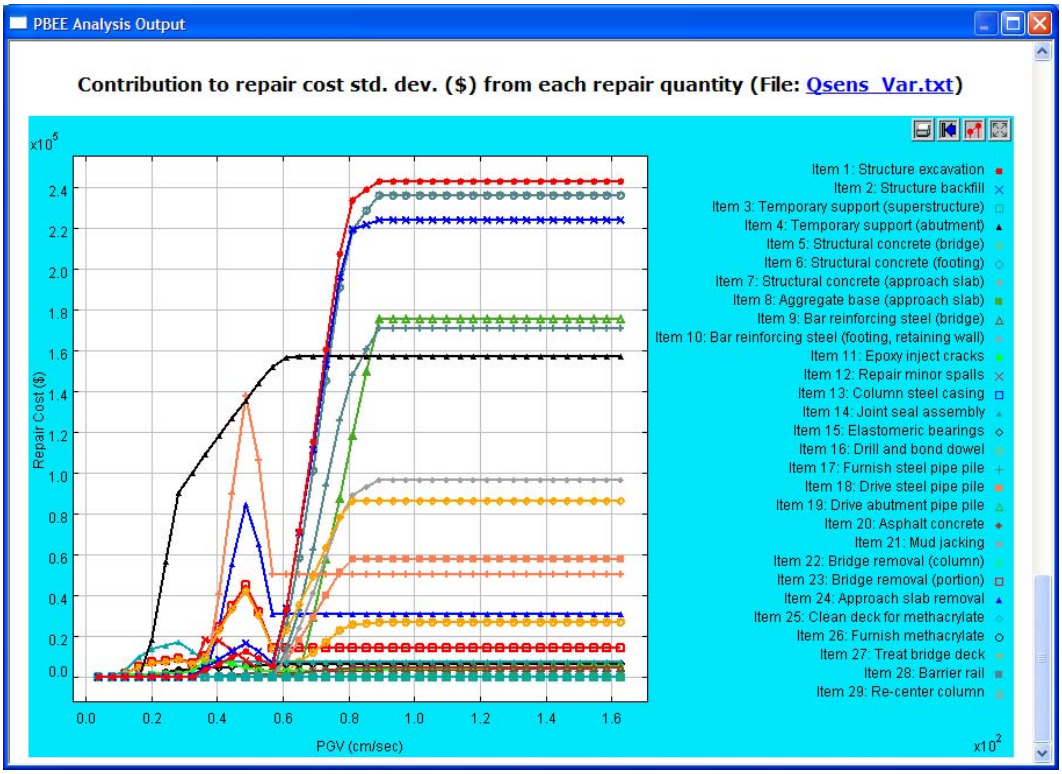

PBEE Analysis Output
Contribution to repair cost std. dev. ($) from each repair quantity (File: Qsens_Var.txt)
x10^5
Repair Cost ($)
PGV (cm/sec)
x10^2
Item 1: Structure excavation
Item 2: Structure backfill
Item 3: Temporary support (superstructure)
Item 4: Temporary support (abutment)
Item 5: Structural concrete (bridge)
Item 6: Structural concrete (footing)
Item 7: Structural concrete (approach slab)
Item 8: Aggregate base (approach slab)
Item 9: Bar reinforcing steel (bridge)
Item 10: Bar reinforcing steel (footing, retaining wall)
Item 11: Epoxy inject cracks
Item 12: Repair minor spalls
Item 13: Column steel casing
Item 14: Joint seal assembly
Item 15: Elastomeric bearings
Item 16: Drill and bond dowel
Item 17: Furnish steel pipe pile
Item 18: Drive steel pipe pile
Item 19: Drive abutment pipe pile
Item 20: Asphalt concrete
Item 21: Mud jacking
Item 22: Bridge removal (column)
Item 23: Bridge removal (portion)
Item 24: Approach slab removal
Item 25: Clean deck for methacrylate
Item 26: Furnish methacrylate
Item 27: Treat bridge deck
Item 28: Barrier rail
Item 29: Re-center column

Fig. 99. Contribution to repair cost standard deviation ($) from each repair quantity

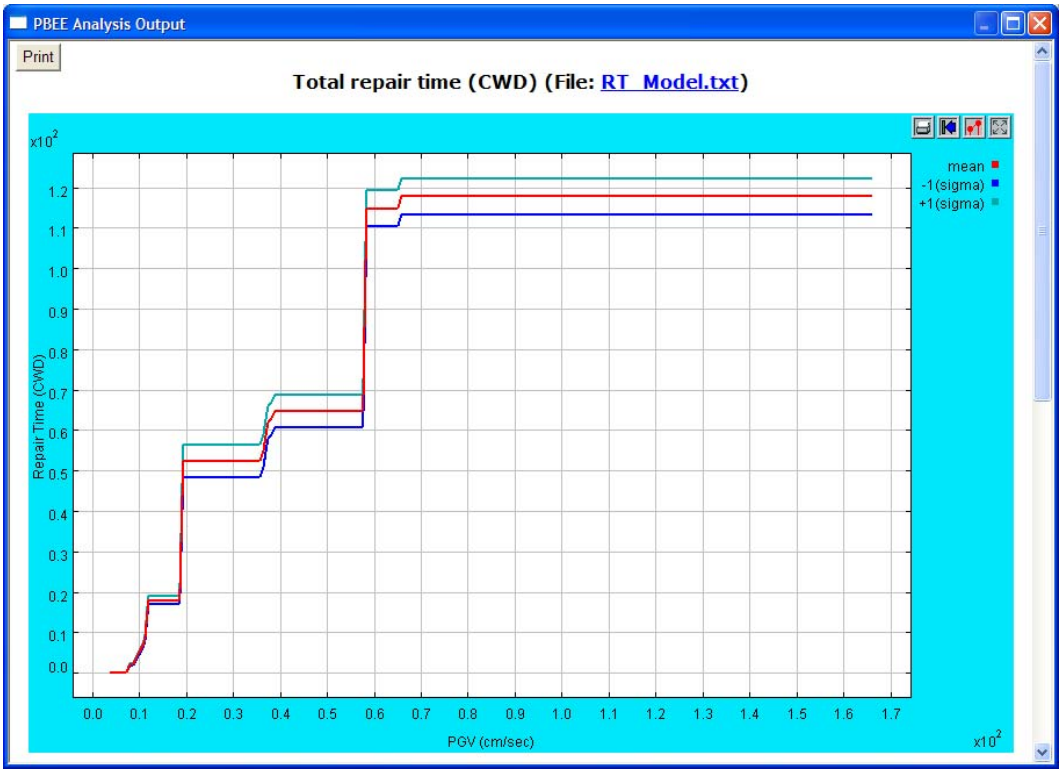


Fig. 100. Total repair time (CWD: Crew Working Day) as a function of intensity

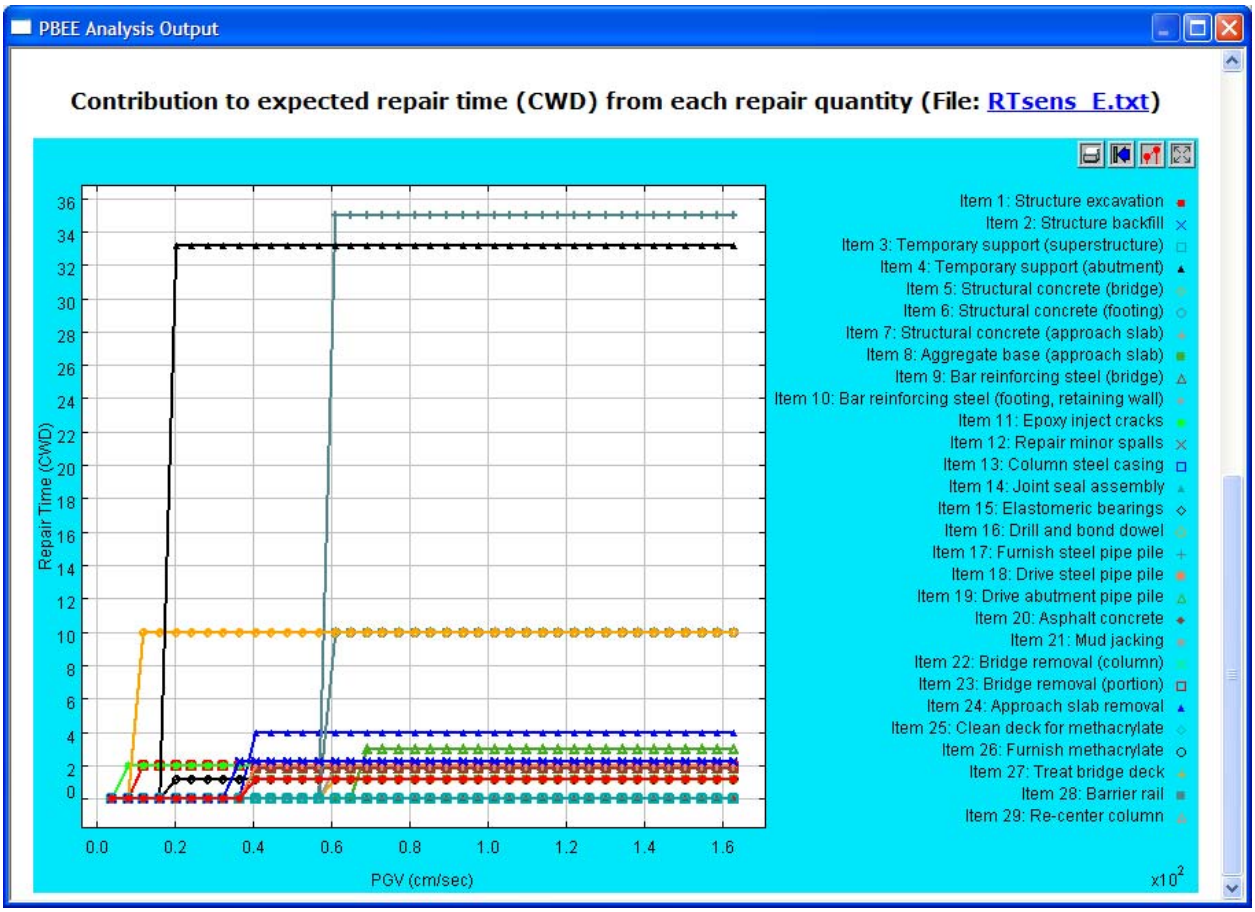

PBEE Analysis Output
Contribution to expected repair time (CWD) from each repair quantity (File: RTsens_E.txt)
Repair Time (CWD)
PGV (cm/sec)
x10^2
Item 1: Structure excavation
Item 2: Structure backfill
Item 3: Temporary support (superstructure)
Item 4: Temporary support (abutment)
Item 5: Structural concrete (bridge)
Item 6: Structural concrete (footing)
Item 7: Structural concrete (approach slab)
Item 8: Aggregate base (approach slab)
Item 9: Bar reinforcing steel (bridge)
Item 10: Bar reinforcing steel (footing, retaining wall)
Item 11: Epoxy inject cracks
Item 12: Repair minor spalls
Item 13: Column steel casing
Item 14: Joint seal assembly
Item 15: Elastomeric bearings
Item 16: Drill and bond dowel
Item 17: Furnish steel pipe pile
Item 18: Drive steel pipe pile
Item 19: Drive abutment pipe pile
Item 20: Asphalt concrete
Item 21: Mud jacking
Item 22: Bridge removal (column)
Item 23: Bridge removal (portion)
Item 24: Approach slab removal
Item 25: Clean deck for methacrylate
Item 26: Furnish methacrylate
Item 27: Treat bridge deck
Item 28: Barrier rail
Item 29: Re-center column

Fig. 101. Contribution to expected repair time (CWD) from each repair quantity

7.2.2 Hazard Curves

Based on the local site Seismic Hazard specified, losses are estimated and displayed graphically as:

- The defined local site hazard curve as a mean annual frequency (ν) of exceedance (ground motion) (Fig. 102)
- Return period against total repair cost ratio (Fig. 103)
- Mean annual frequency (MAF) of exceedance (loss) against total repair cost ratio RCR (Fig. 104)
- Return period against total repair time RT (Fig. 105)
- Mean annual frequency (MAF) of exceedance (loss) against total repair time (Fig. 106)

The median ground motion hazard curve is assumed to have a power-law form with two unknown parameters (k, k_0 in Eq. 4) in the range of the ground motion intensities bracketed by the 2%- and 10%-probability of exceedance IM values (im). The two-parameter fit (linear in log

space) to the nonlinear (in log space) hazard curve tends to overpredict frequencies of exceedance for IM extremes both above and below the range of intensities considered. Therefore, care should be taken when extrapolating any resultant hazard curves to extremely low (or high) frequencies of exceedance. Using a least-squares fit in log space, the unknown parameters can be determined numerically from the three values input by the user (2%-, 5%-, and 10%-probability of exceedance in 50 years). On the site hazard curves plotted in the interface, both the data points and the fitted curve are shown (Fig. 102).

$$\nu_{IM}(im) = k_0(im)^{-k} \tag{4}$$

The power-law fit to the hazard data is used to compute the loss hazards. The loss model (probability of exceeding RCR or RT conditioned on intensity levels) is integrated with the absolute value of the derivative of this IM hazard to obtain the loss hazard curve (MAF of exceeding either RCR or RT). Details of the numerical integration are presented in Mackie et al. (2008) and other sources.

The loss hazard curves (both for repair cost and repair time) are further integrated over intensity to yield mean annual loss. For example, in Fig. 104, the mean annual repair cost ratio expected for the bridge at the given site is 0.05% of the replacement cost.

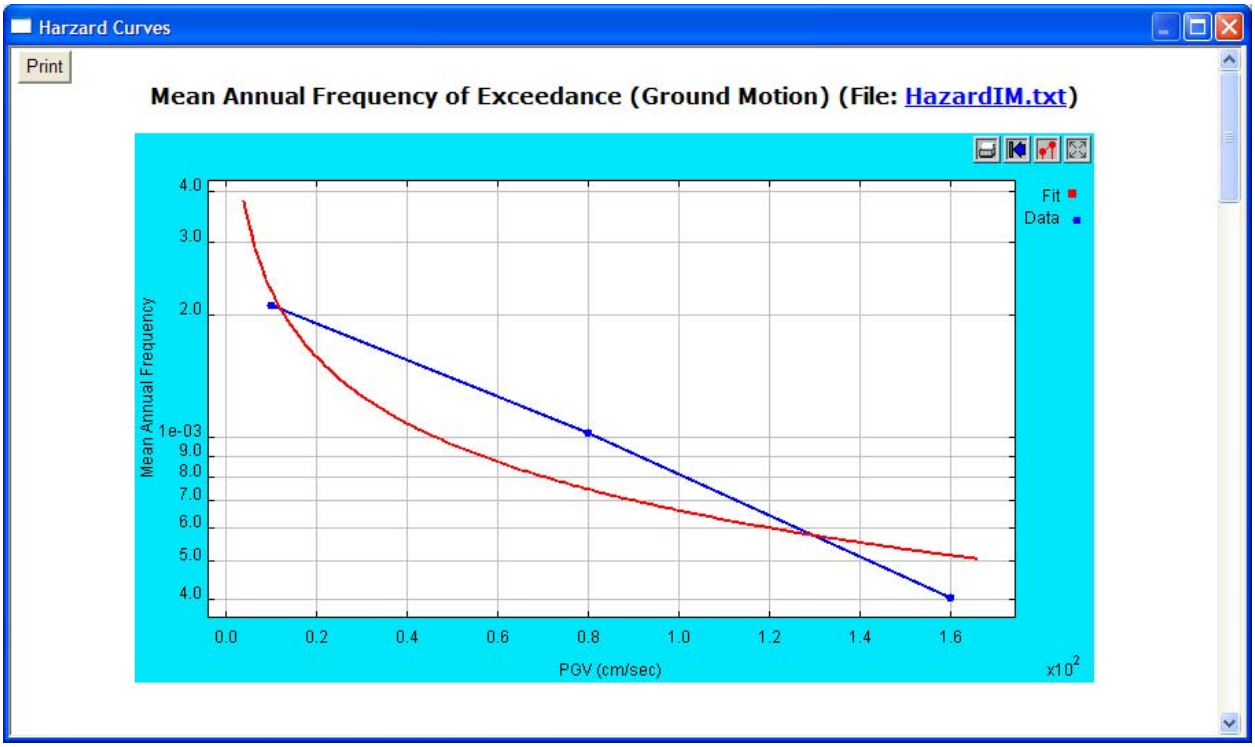


Fig. 102. Mean annual frequency of exceedance (ground motion)

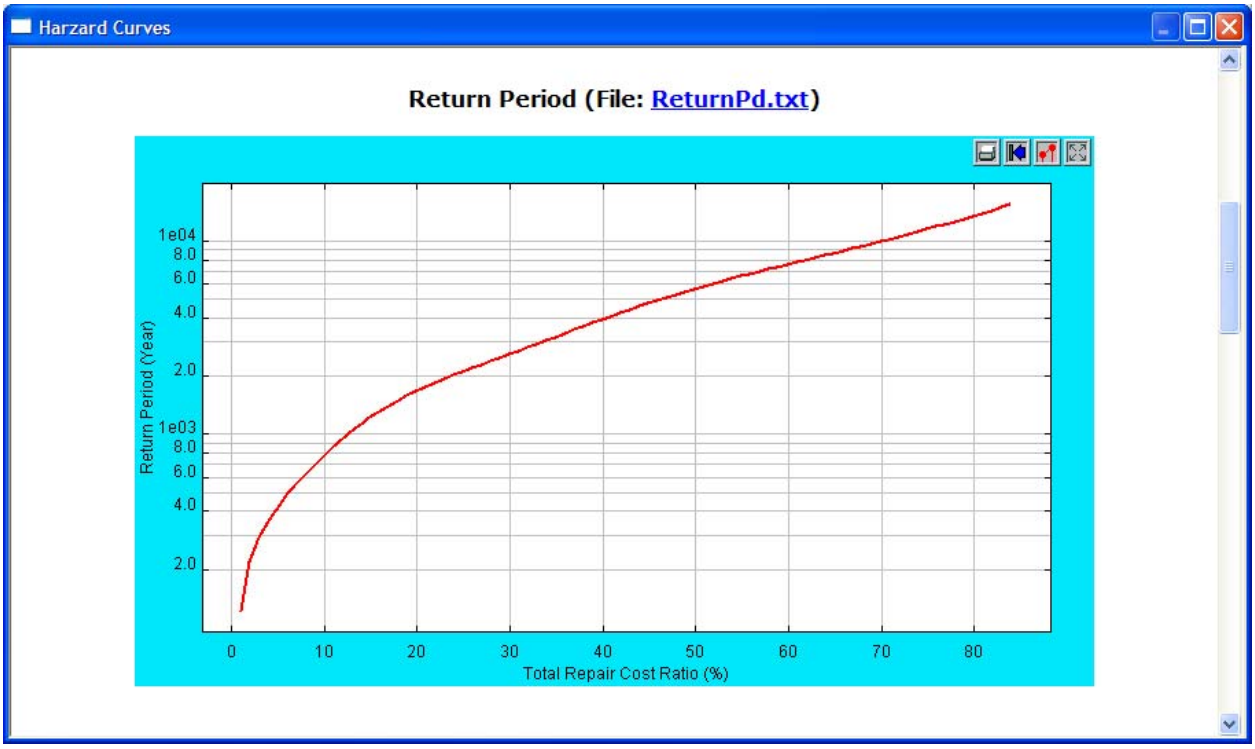


Fig. 103. Return period against total repair cost ratio

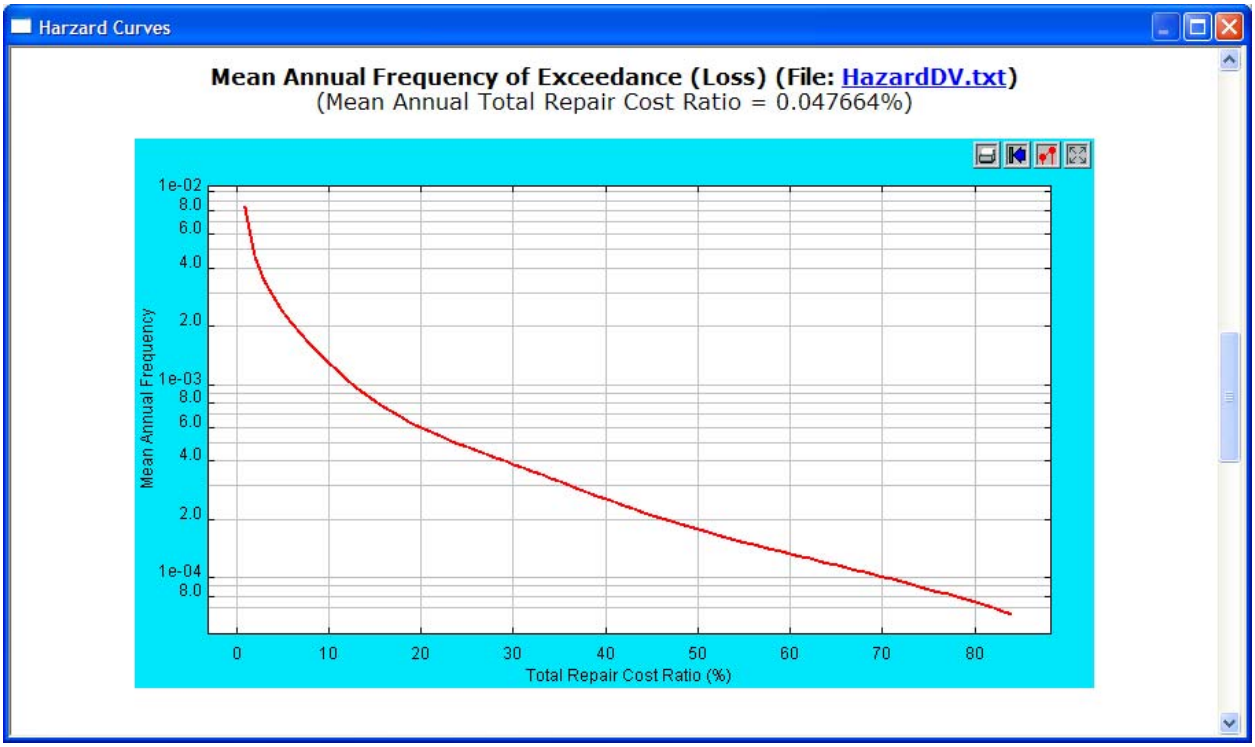


Fig. 104. Mean annual frequency of exceedance (loss) against total repair cost ratio

Fig. 105. Return period against total repair time

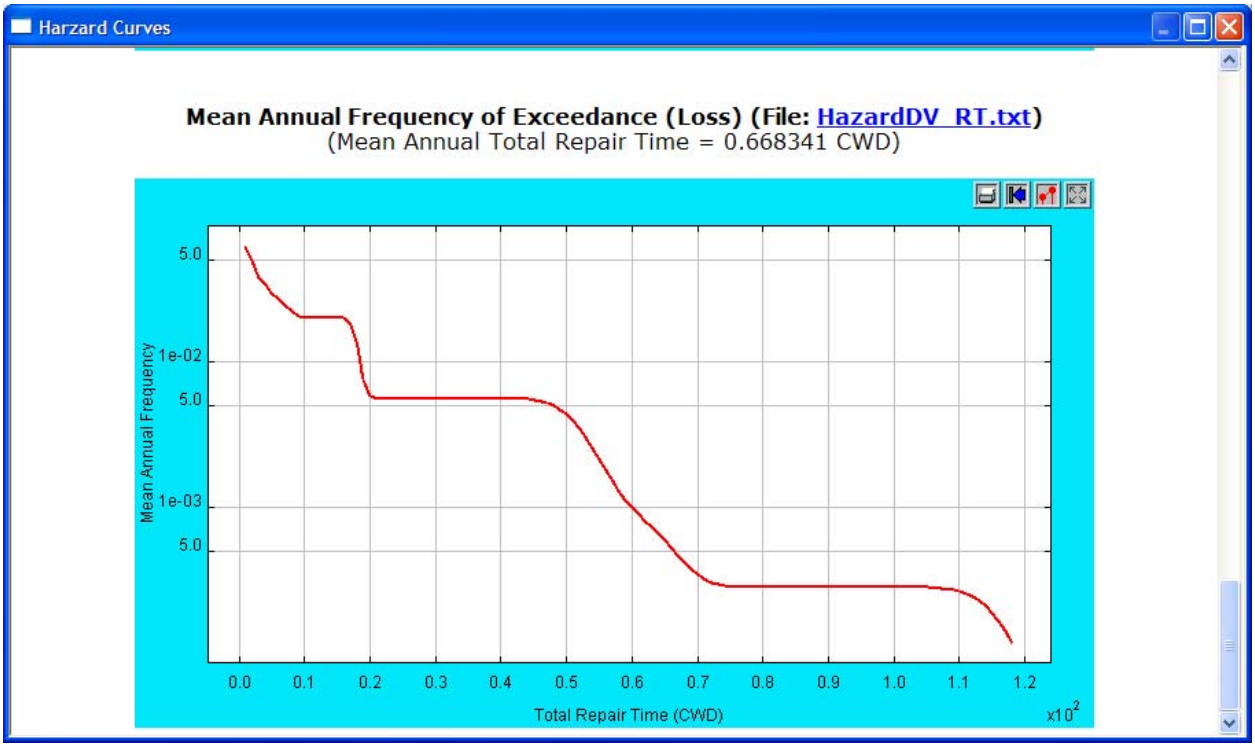


Fig. 106. Mean annual frequency of exceedance (loss) against total repair time

7.2.3 Disaggregation

Figs. 107-109 display the disaggregation (Fig. 57) of expected cost by performance group, the disaggregation of expected cost by repair quantities, and the disaggregation of expected time by repair quantities, respectively. In thie figures below, the disaggregation is performed at an intensity of 100 cm/s (PGV) for all three figures (a user IM and value as shown in Fig. 57). This IM and its value are shown in the plot titles.

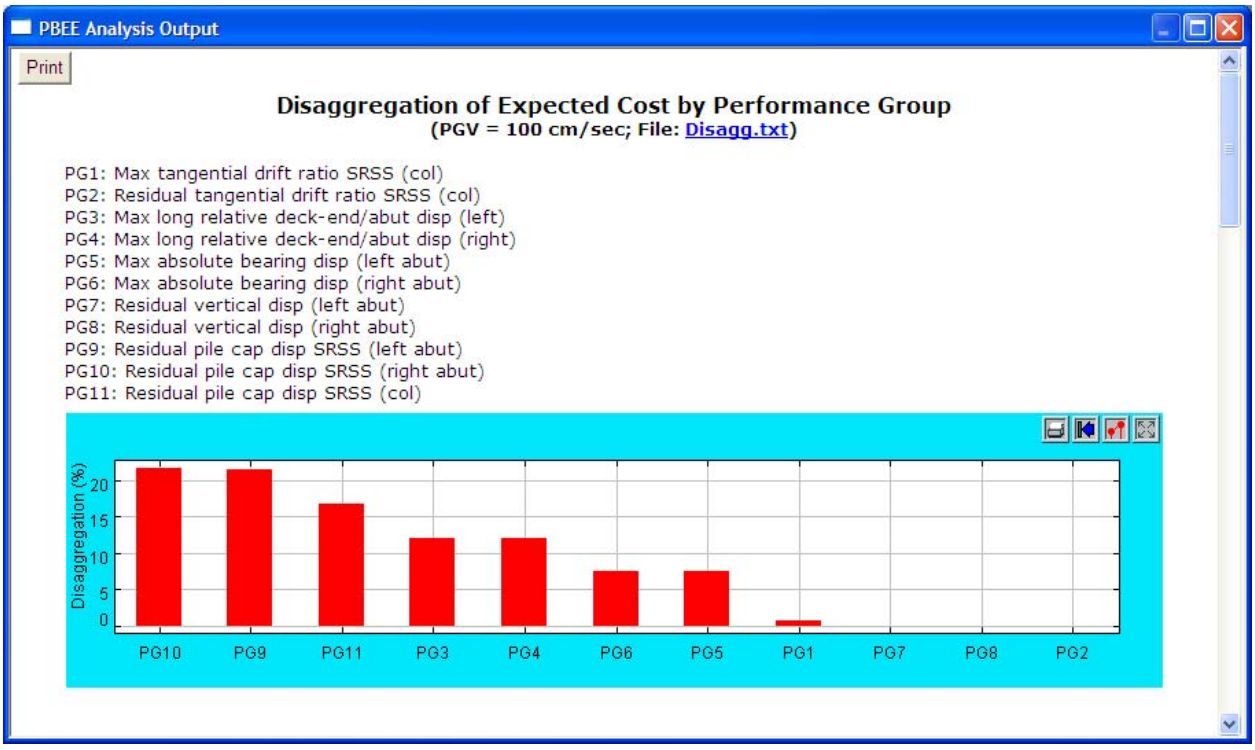


Fig. 107. Disaggregation of expected cost by performance group

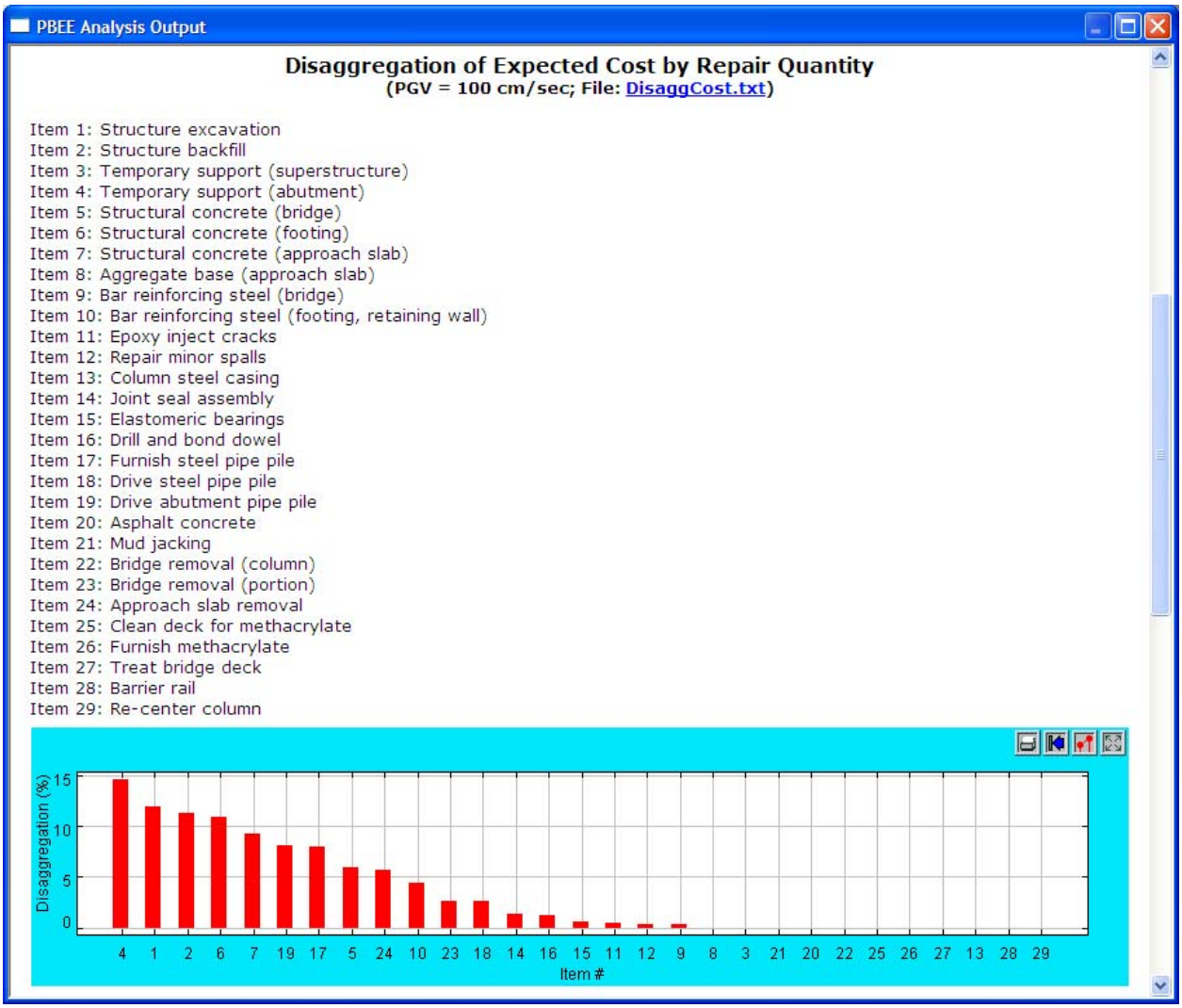

PBEE Analysis Output
Disaggregation of Expected Cost by Repair Quantity
(PGV = 100 cm/sec; File: DisaggCost.txt)
Item 1: Structure excavation
Item 2: Structure backfill
Item 3: Temporary support (superstructure)
Item 4: Temporary support (abutment)
Item 5: Structural concrete (bridge)
Item 6: Structural concrete (footing)
Item 7: Structural concrete (approach slab)
Item 8: Aggregate base (approach slab)
Item 9: Bar reinforcing steel (bridge)
Item 10: Bar reinforcing steel (footing, retaining wall)
Item 11: Epoxy inject cracks
Item 12: Repair minor spalls
Item 13: Column steel casing
Item 14: Joint seal assembly
Item 15: Elastomeric bearings
Item 16: Drill and bond dowel
Item 17: Furnish steel pipe pile
Item 18: Drive steel pipe pile
Item 19: Drive abutment pipe pile
Item 20: Asphalt concrete
Item 21: Mud jacking
Item 22: Bridge removal (column)
Item 23: Bridge removal (portion)
Item 24: Approach slab removal
Item 25: Clean deck for methacrylate
Item 26: Furnish methacrylate
Item 27: Treat bridge deck
Item 28: Barrier rail
Item 29: Re-center column
Disaggregation (%)
0
5
10
15
4 1 2 6 7 19 17 5 24 10 23 18 14 16 15 11 12 9 8 3 21 20 22 25 26 27 13 28 29
Item #

Fig. 108. Disaggregation of expected repair cost by repair quantities

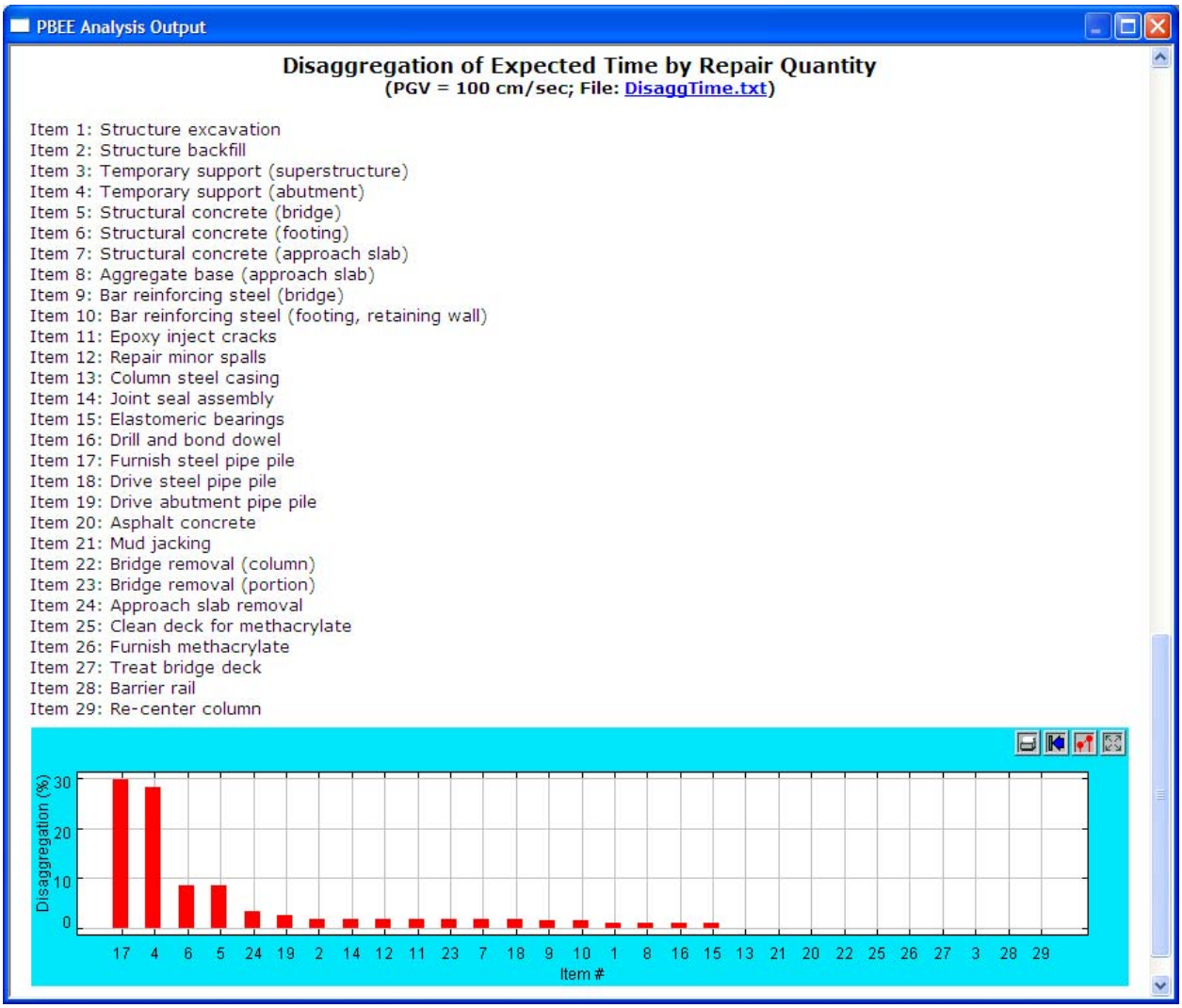

PBEE Analysis Output
Disaggregation of Expected Time by Repair Quantity
(PGV = 100 cm/sec; File: DisaggTime.txt)
Item 1: Structure excavation
Item 2: Structure backfill
Item 3: Temporary support (superstructure)
Item 4: Temporary support (abutment)
Item 5: Structural concrete (bridge)
Item 6: Structural concrete (footing)
Item 7: Structural concrete (approach slab)
Item 8: Aggregate base (approach slab)
Item 9: Bar reinforcing steel (bridge)
Item 10: Bar reinforcing steel (footing, retaining wall)
Item 11: Epoxy inject cracks
Item 12: Repair minor spalls
Item 13: Column steel casing
Item 14: Joint seal assembly
Item 15: Elastomeric bearings
Item 16: Drill and bond dowel
Item 17: Furnish steel pipe pile
Item 18: Drive steel pipe pile
Item 19: Drive abutment pipe pile
Item 20: Asphalt concrete
Item 21: Mud jacking
Item 22: Bridge removal (column)
Item 23: Bridge removal (portion)
Item 24: Approach slab removal
Item 25: Clean deck for methacrylate
Item 26: Furnish methacrylate
Item 27: Treat bridge deck
Item 28: Barrier rail
Item 29: Re-center column
Disaggregation (%)
30
20
10
0
17 4 6 5 24 19 2 14 12 11 23 7 18 9 10 1 8 16 15 13 21 20 22 25 26 27 3 28 29
Item #

Fig. 109. Disaggregation of expected repair time by repair quantities

7.2.4 EPS Version of All PBEE Figures

A MS Word file contained all PBEE figures in the EPS format can be created as shown below.

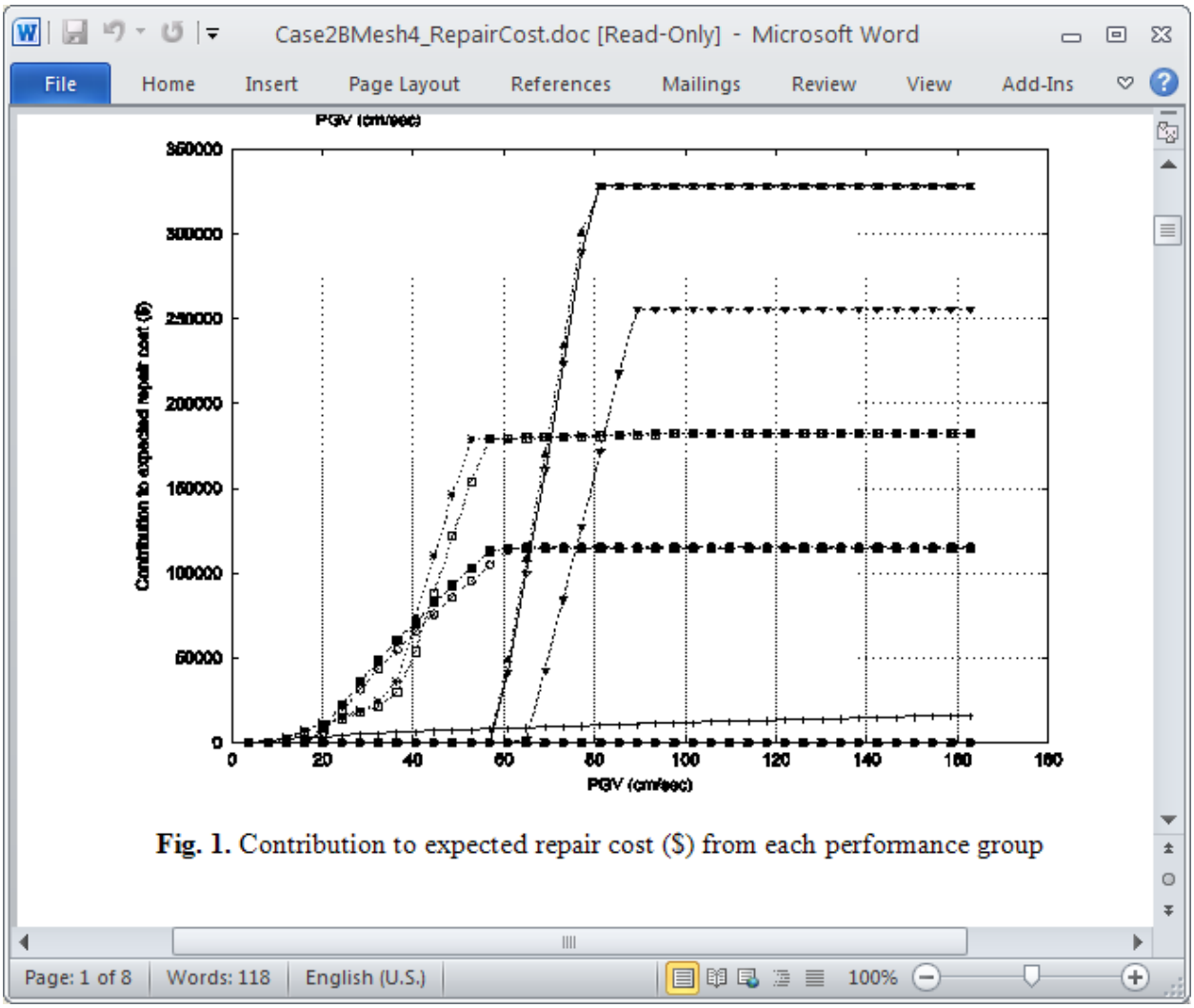
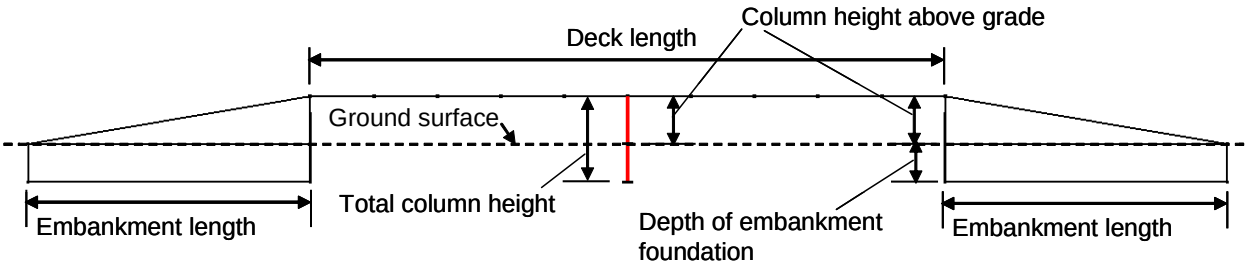


Fig. 1. Contribution to expected repair cost ($) from each performance group

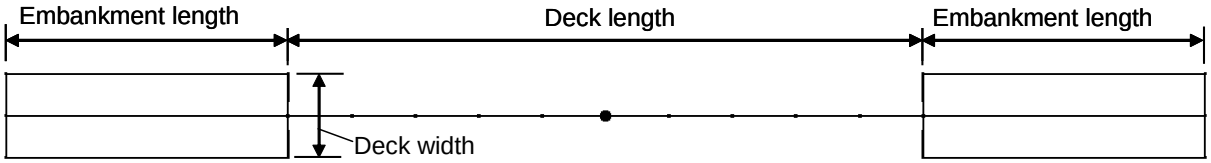
Fig. 110. Converting all PBEE figures to EPS format

8 Appendix A: How to Define the Soil Finite Element Mesh

A bridge and approach embankments supported on a ground strata will be defined. The bridge configuration is shown below (Fig. 111). In this simple configuration, the approach embankments are idealized by a rigid triangular configuration employed to exert the self-weight of these embankments on the supporting ground.



Side view



Plan view

Fig. 111. Schematic view of an idealized single bent bridge system

Step 1

In the user interface, click **Bridge Parameters**. With reference to Fig. 111, define the following parameters according to your preference:

Diameter: This is the bridge column outer diameter, which is currently also the pile diameter (Integral column foundation scenario).

Total Column Length: Starting from the bridge deck all the way to the pile tip

Column Length above Surface: from bridge deck to mud-line

Embankment Length: (in plan view, longitudinally from bridge edge to street level away from bridge)

Depth of Embankment Foundation: Height of approach embankment at bridge edge from the ground surface to the base of the approach embankment foundation (Fig. 111).

Deck Length: Length of bridge in the longitudinal direction

Soil Parameters: make sure at least the total "**Thickness**" of soil layers is defined: This is the total thickness of the ground stratum from the ground surface all the way down to the base of the soil mesh. Make sure that the column/pile base (tip) is within the defined soil domain depth. Note: Earthquake input motion is imparted along the base of the soil mesh. This base is assumed to represent rigid bedrock. As such, this input earthquake excitation constitutes total motion imparted at this Bedrock level.

Step 2
Click **Mesh Parameters** to define additional meshing parameters.

Tab "**General Definition**" (Fig. 112): Make sure "**Mesh Scale**" is "Full mesh" and "**Number of Slices**" is 16 or larger. This parameter refines the mesh by creating additional elements in horizontal plane of the soil mesh.

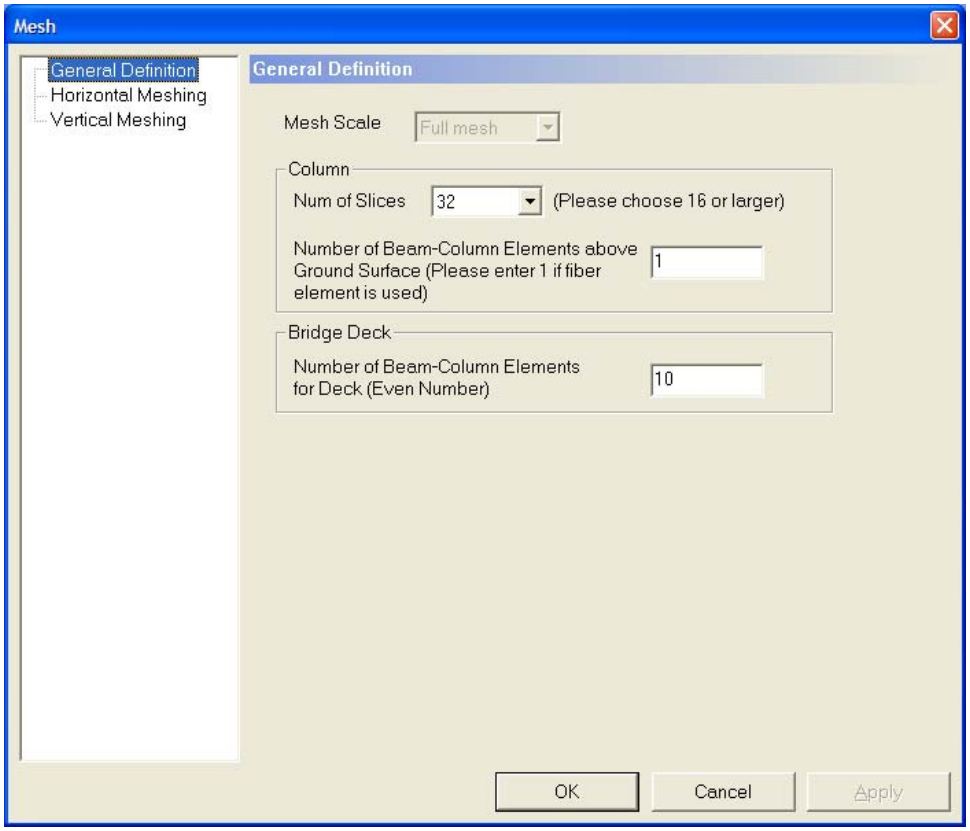


Fig. 112. General meshing controlling parameters (default values)

Tab "**Horizontal Meshing**" (Fig. 113): This section controls mesh refinement along the horizontal direction. **Length** of each soil horizontal layer is defined in the left column. Number

of mesh elements in each defined is specified in the column "**Number of Mesh Layers**". Note that the first mesh layer is starting from the center of the mesh when the column is located and the length of the first mesh layer is equal to the column radius. The fourth mesh layer is for the embankment. **Ratio of Element Length over Next** is used to obtain a gradually changing element size within a layer if **Uniform Meshing** is unchecked (obviously this option is only valid if the # of mesh layers is 2 or larger).

To obtain a refinement near the embankment (Fig. 114), check **Activate Adjusting Mesh beside Embankment** (this option is only valid if **Num of Slices** is 32 or larger). Then define the size of the 1st layer as a factor of Deck Width. Fig. 114 shows an example of using this option.

To minimize the number of elements in the horizontal direction, check **Minimize Number of Elements in Horizontal Direction**. As a result, all other options defined for the horizontal meshing will be ignored. This checkbox option is particularly useful when a rigid ground case is needed.

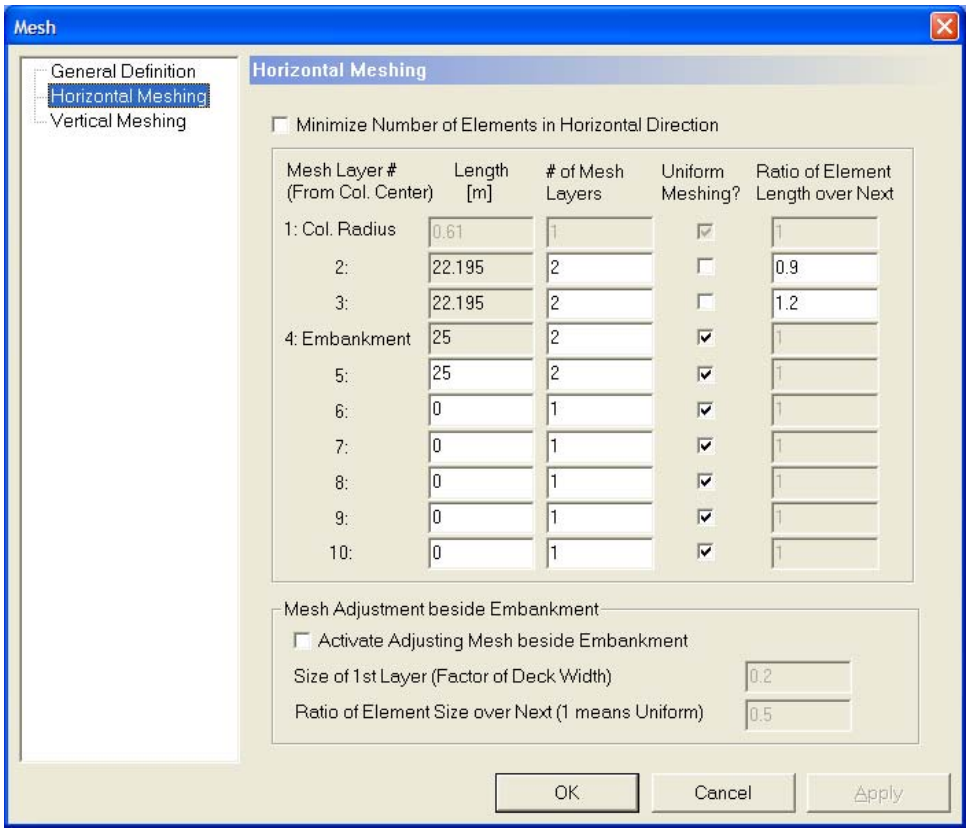
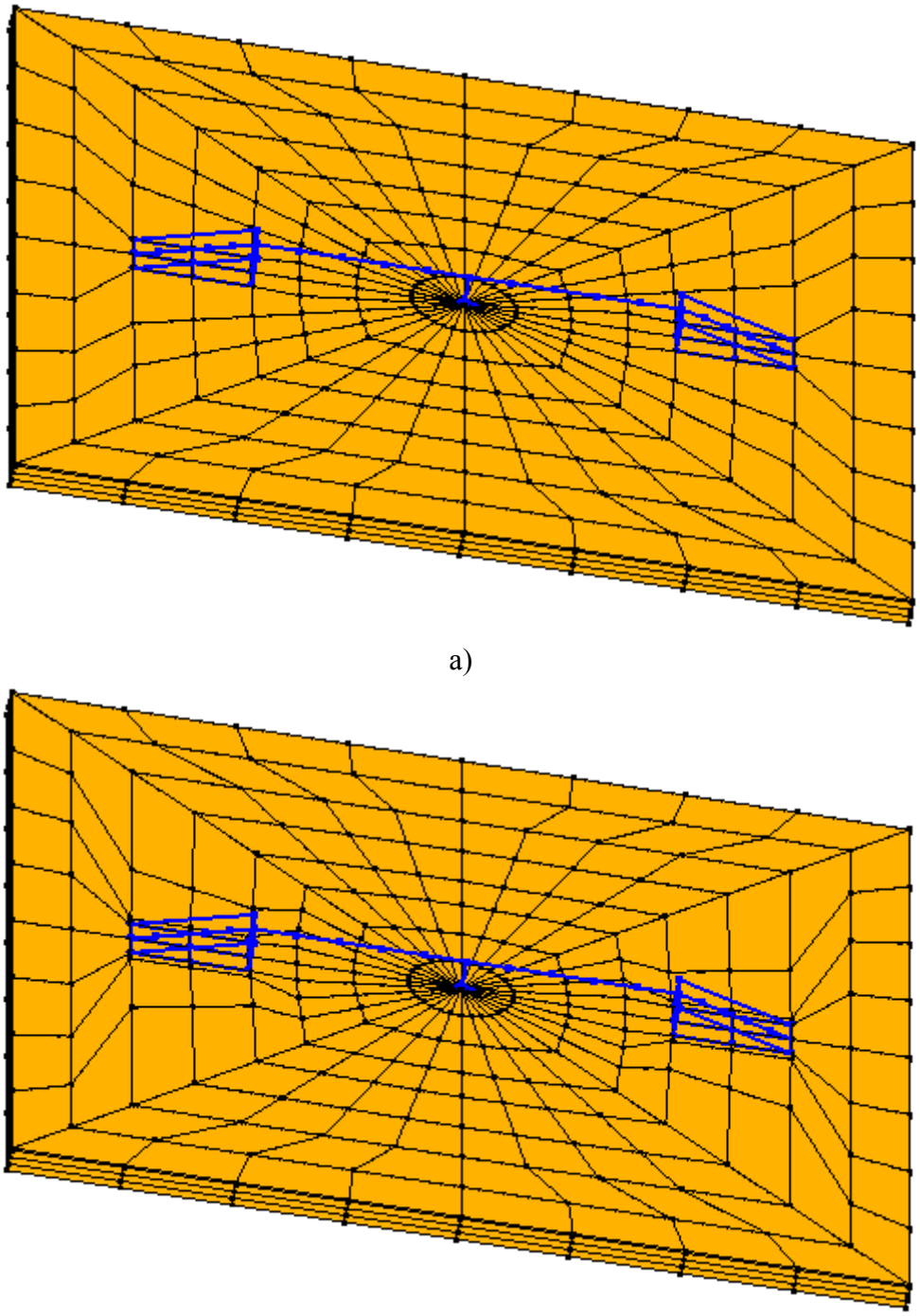


Fig. 113. Meshing controlling parameters for horizontal direction (default values)



a)

b)

Fig. 114. Adjusting mesh near embankment: a) before adjusting; b) after adjusting

Tab "**Vertical Meshing**" (Fig. 115): This section defines the soil profile (layering) along the vertical direction starting from the ground surface downwards (looking at the side view from the top downwards. **Height** (thickness) of each soil layer is defined in the left column. Number of mesh elements in each defined is specified in the column "**Number of Mesh Layers**" (at least

equal to 1 to define a soil profile consisting of a single type of soil). Height (thickness) of this layer must be equal to the entire soil stratum height. Note that the number of mesh layers in the upper zone (where the pile foundation is embedded) will automatically define the number of beam column elements of this pile (below ground surface). As such, it is generally advisable to select an adequate number of mesh layers in this zone. Note: If there is any error during mesh generation, please follow the error message instructions to adjust the controlling parameters and then try again.

Note: Element size is a parameter that affects frequency content of the ground response. Smaller size elements (particularly along the soil domain height), will permit higher frequencies (if present in the input motion) to propagate to the ground surface with more fidelity.

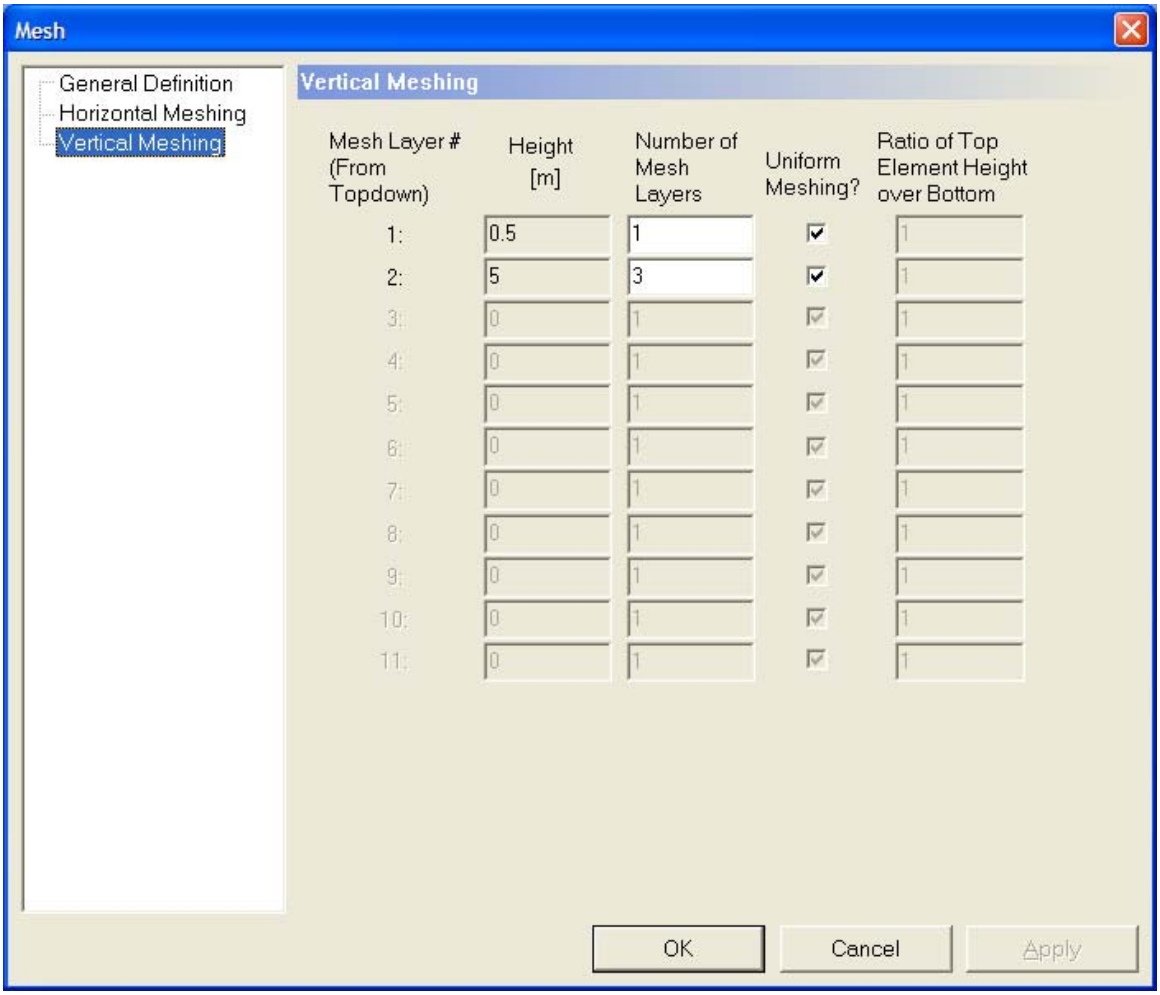


Fig. 115. Meshing controlling parameters for vertical direction (default values)

The finite element mesh created with the above default values is shown in Fig. 116. Examples of mesh generation are shown in Figs. 117-119.

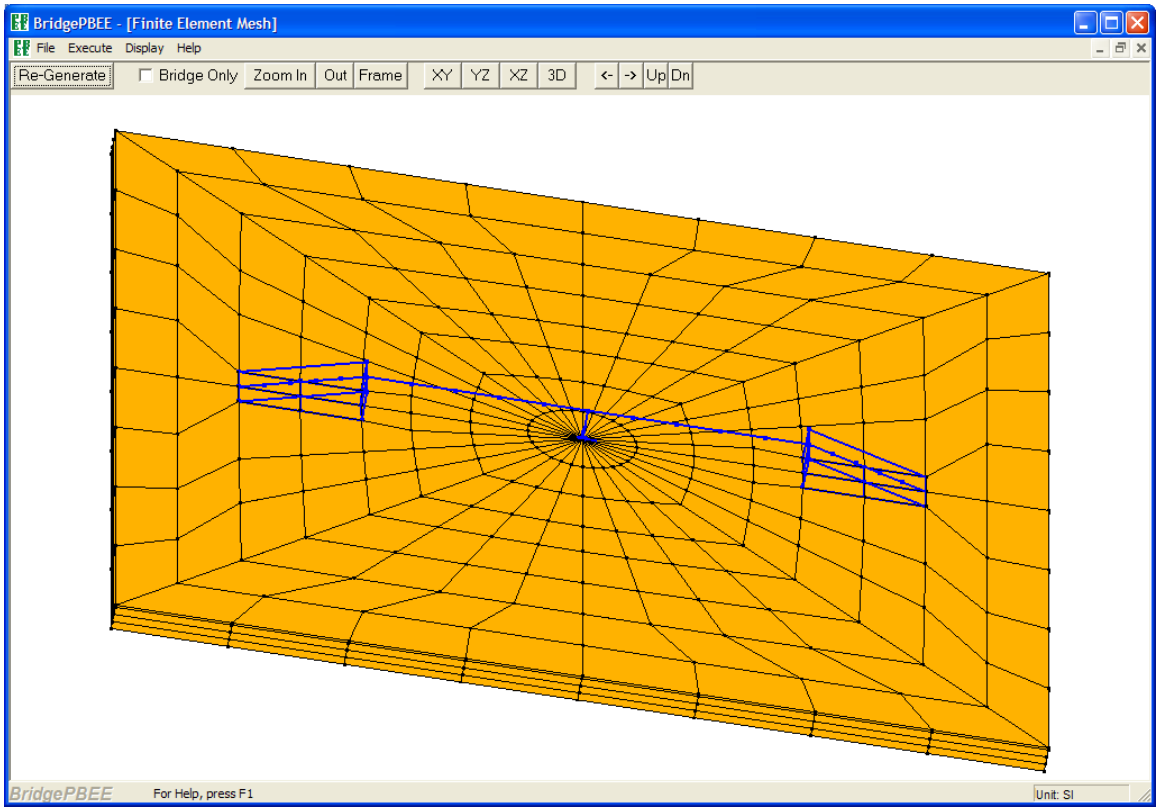
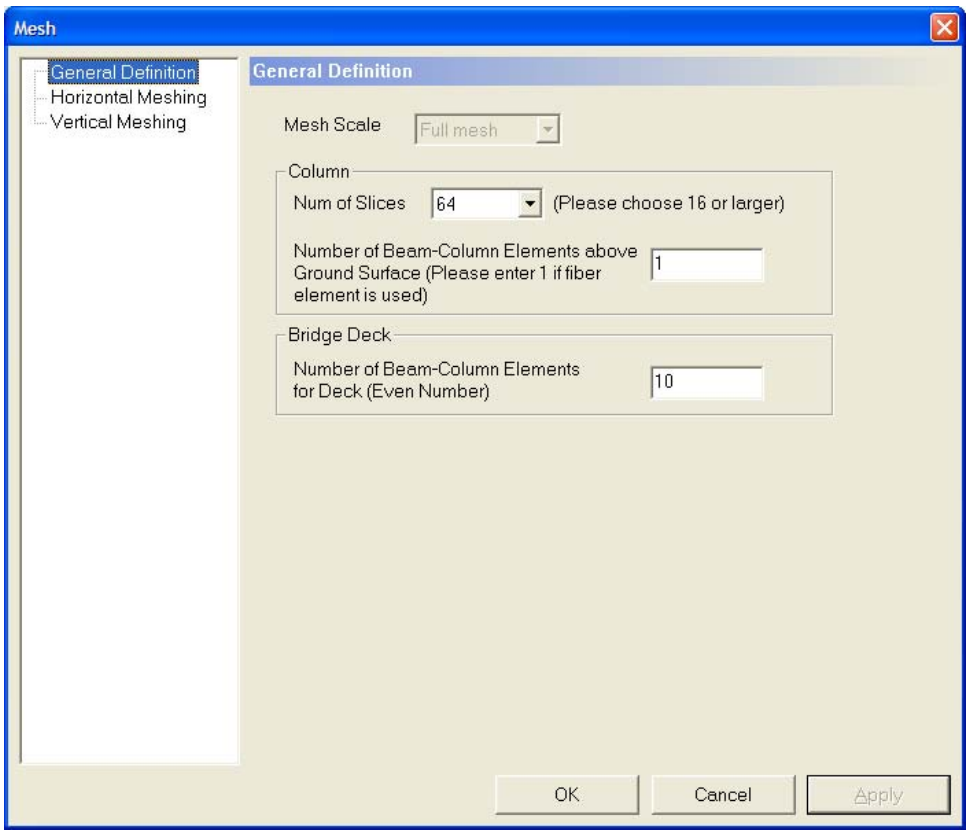
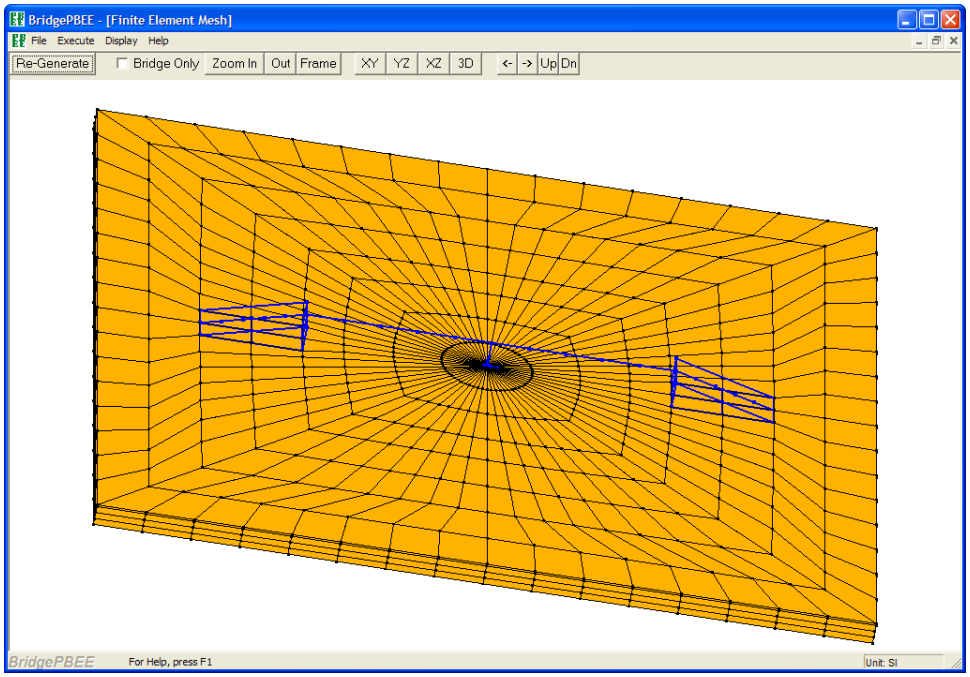


Fig. 116. Finite element mesh created with default values

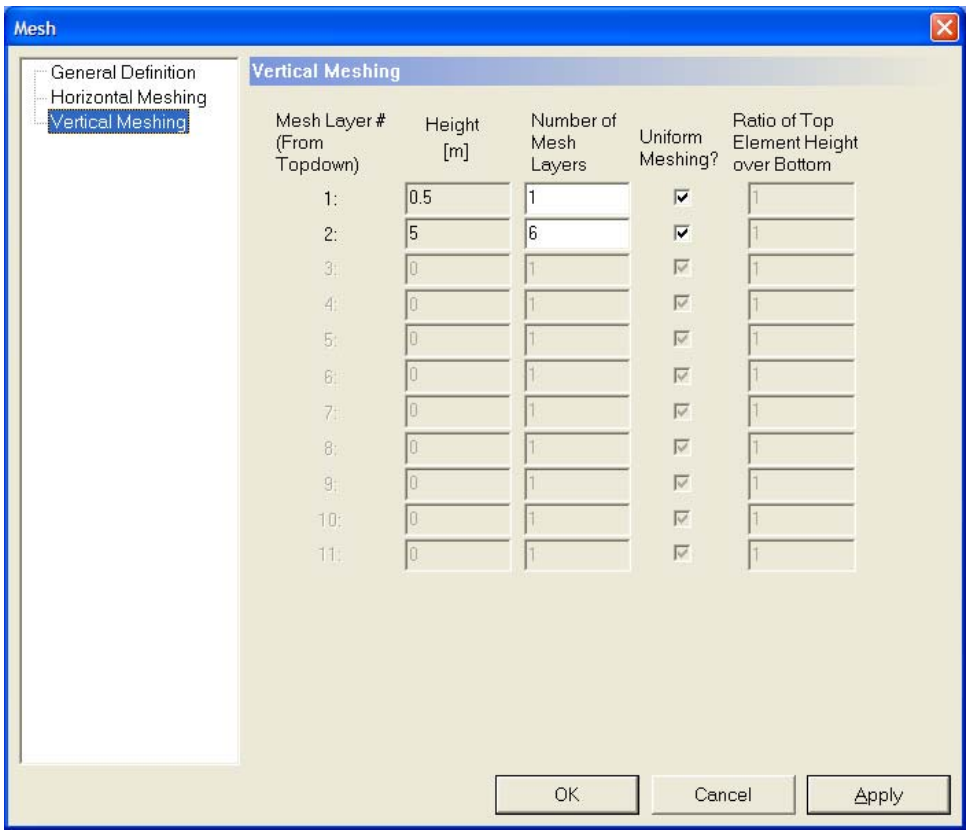


a)

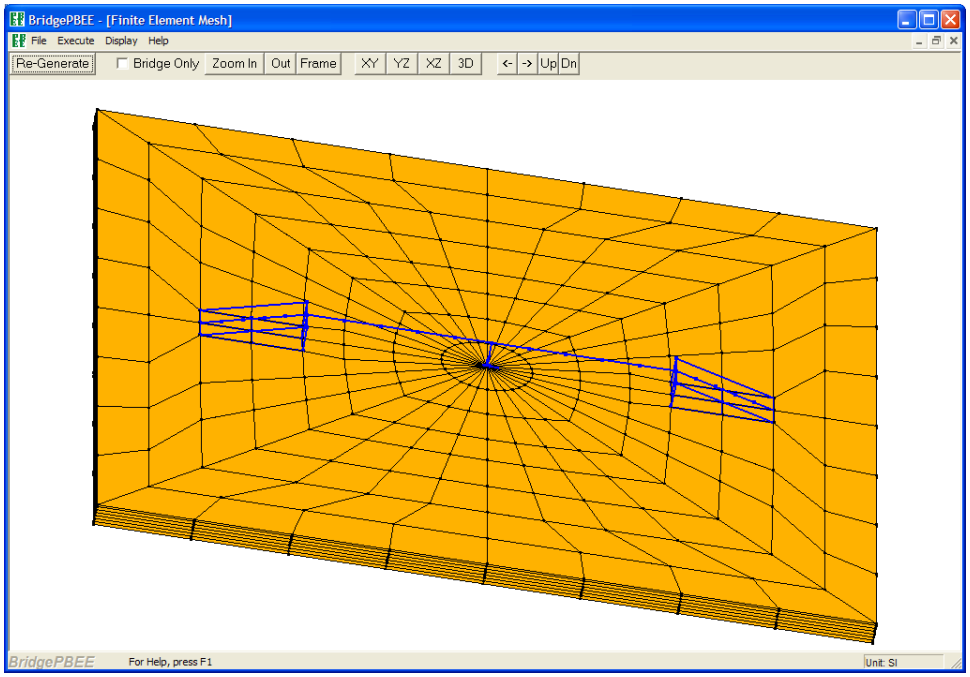


b)

Fig. 117. Mesh refinement example 1: a) Change "Num of Slices" to 32; b) the resulting mesh

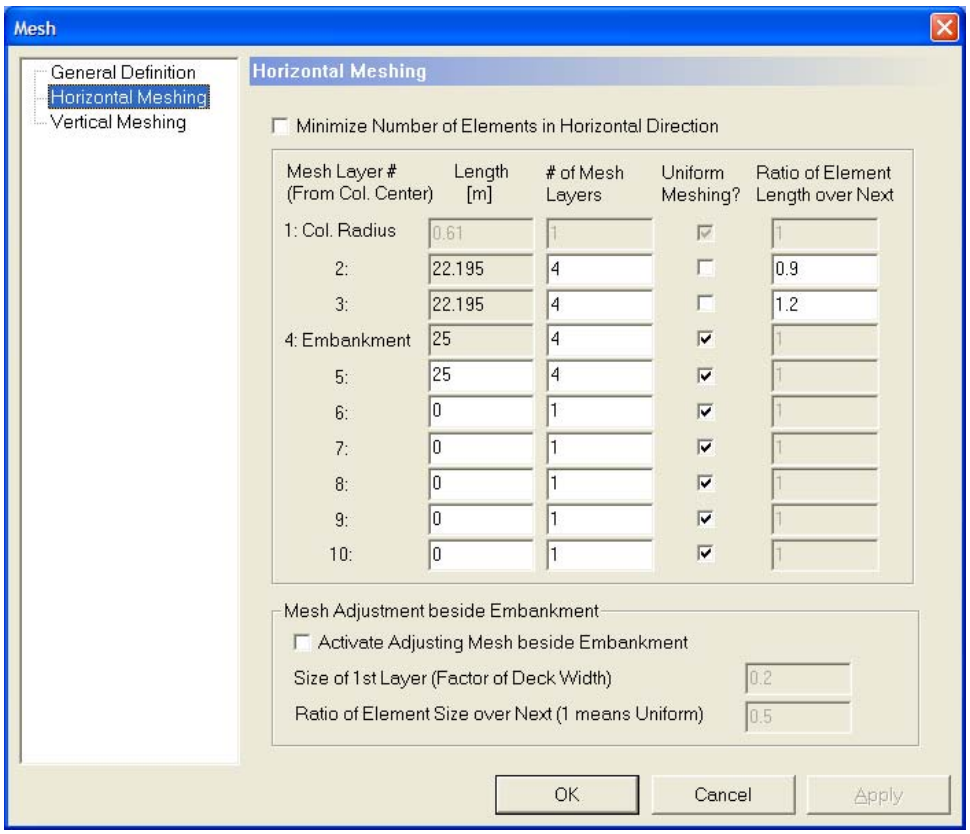


a)

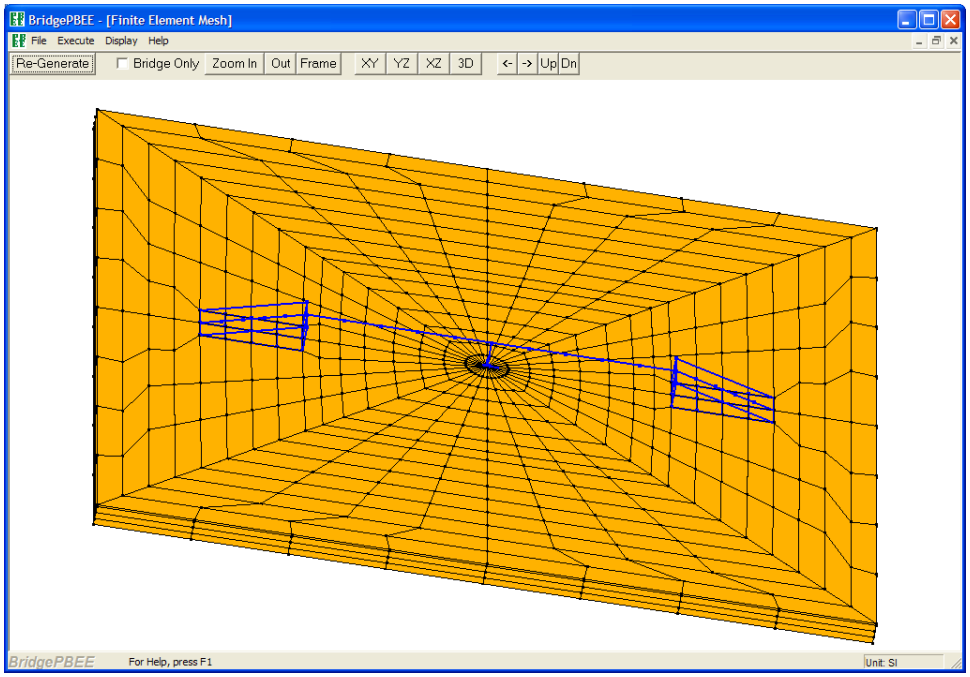


b)

Fig. 118. Mesh refinement example 2: a) Change "Number of Mesh Layers" in the vertical direction; b) the resulting mesh



a)



b)

Fig. 119. Mesh refinement example 3: a) Change meshing controlling parameters in the horizontal direction; b) the resulting mesh

9 Appendix B: Simple Pushover Examples (Bridge on Rigid Ground)

Steps to build a bridge model on a fixed base

In BridgePBEE, a very stiff ground mesh is currently used to simulate a fixed-base scenario.

Simplest Approach:
Start with Example 1 at http://peer.berkeley.edu/bridgepbee/ and modify the bridge model to match your specifications.

Alternative Approach:
To make the soil very stiff, please follow the steps below:
Step 1: In the main page of the interface (Fig. 2), click **File**, then **New Model**, and **Yes** (to create a new model).
Step 2: Click **Soil Parameters** in the main window.
Step 3: Click **22: U-Clay2** from **Soil Type** dropdown list.
Step 4: Enter a large number for the **Shear Wave Velocity** (e.g., 10,000 m/sec), and click **OK** to close U-Clay2 window.
Step 5: Click **OK** to close **Soil Strata** window.
Step 6: Click **File**, and then **Save Model** (to save the model).

Simple Verification (Linear column properties)

1. Cantilever Beam with Longitudinal Load at Free End

This case can be obtained by making the bridge deck very flexible (e.g., use a very small value for the elastic modulus). The Roller abutment model is employed.

The Fiber section with Elastic Material is used to simulate the column. In this case, the equivalent flexural stiffness is EI = 3375450 kN-m^2 (as reported back by the user interface, see Fig. 10, when "Elastic" is selected).

Load P = 20 kN
Length L = 6.71 m

The end displacement w = PL^3/3EI = 5.97 E-04 m

BridgePBEE gives 6.1E-04 m (Fig. 120).

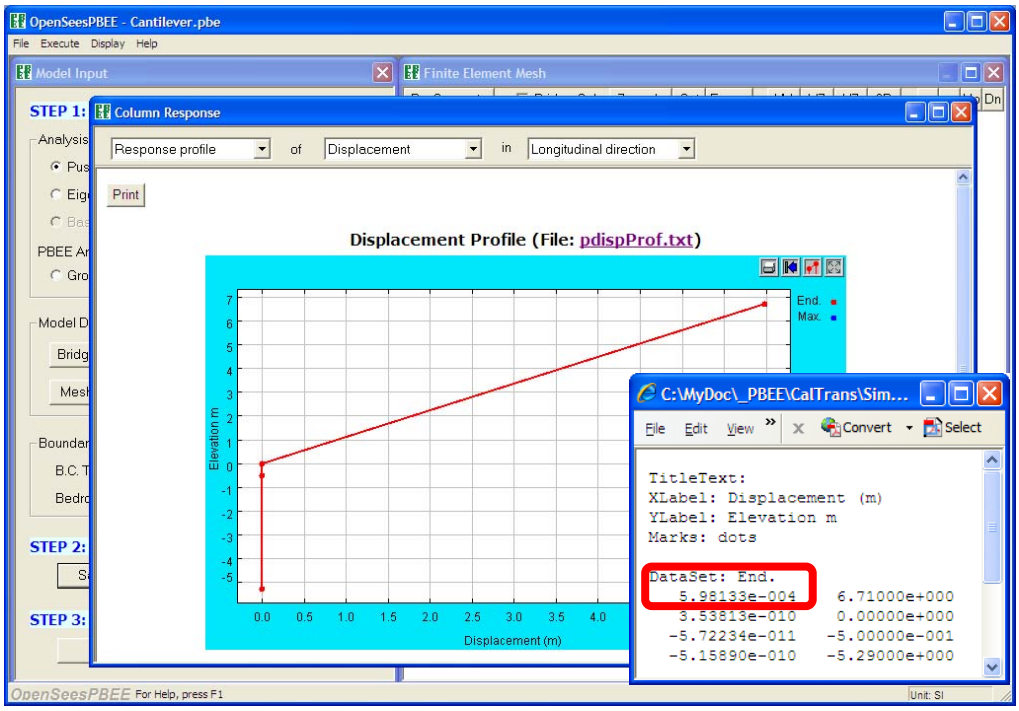


Fig. 120. Cantilever beam simulation using BridgePBEE

2. Fixed-end Beam with Point Load

This is a case where the column base is fixed at rigid rock and there is zero rotation at the column top. This case can be obtained by making the bridge deck very stiff and also applying the Roller abutment model.

The Fiber section with Elastic material is used to simulate the column. In this case, the equivalent flexural stiffness EI = 3375450 kN-m^2 (as reported by the user interface, when "Elastic" properties are selected in Fig. 10).

Load P = 40 kN (half load 20 kN is used in the user interface)
Length L = 13.42 m (half length 6.71 m is used in the user interface)

The end displacement $w = PL^3/12EI = 1.492$ E-04 m

BridgePBEE gives 1.496E-04 m (Fig. 121).

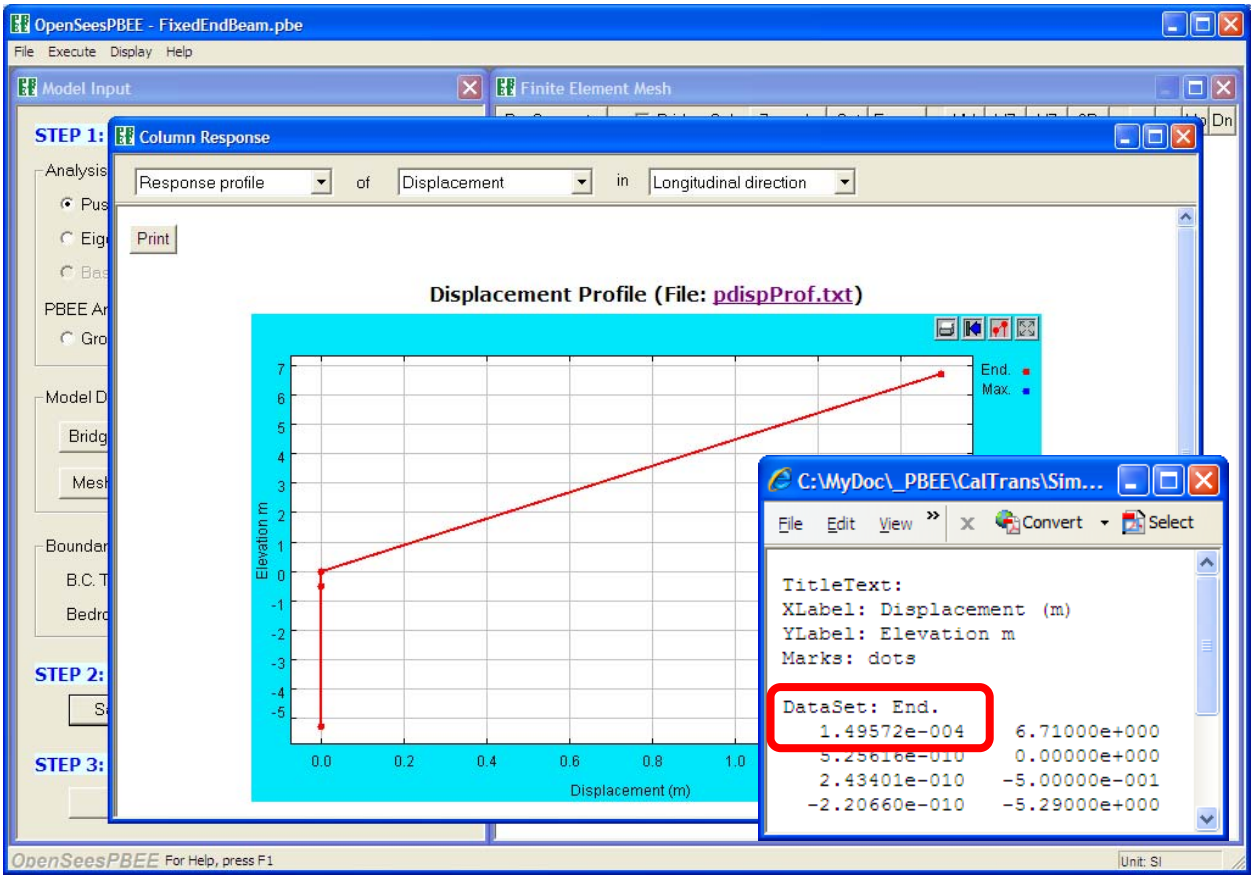


Fig. 121. Fixed-end beam simulation using BridgePBEE

3. Bridge self-weight with rigid Column & roller abutment model

The distributed load of deck p = 130.3 kN/m
Half of the bridge length L = 45 m
Elastic modulus of deck = 28,000,000 kPa
Moment of Inertia = 2.81 m^4

Fig. 122 displays the deformation of bridge deck under gravity. The maximum is 0.0372 m (Fig. 122). The close-form solution gives 0.0368 m (Fig. 123).

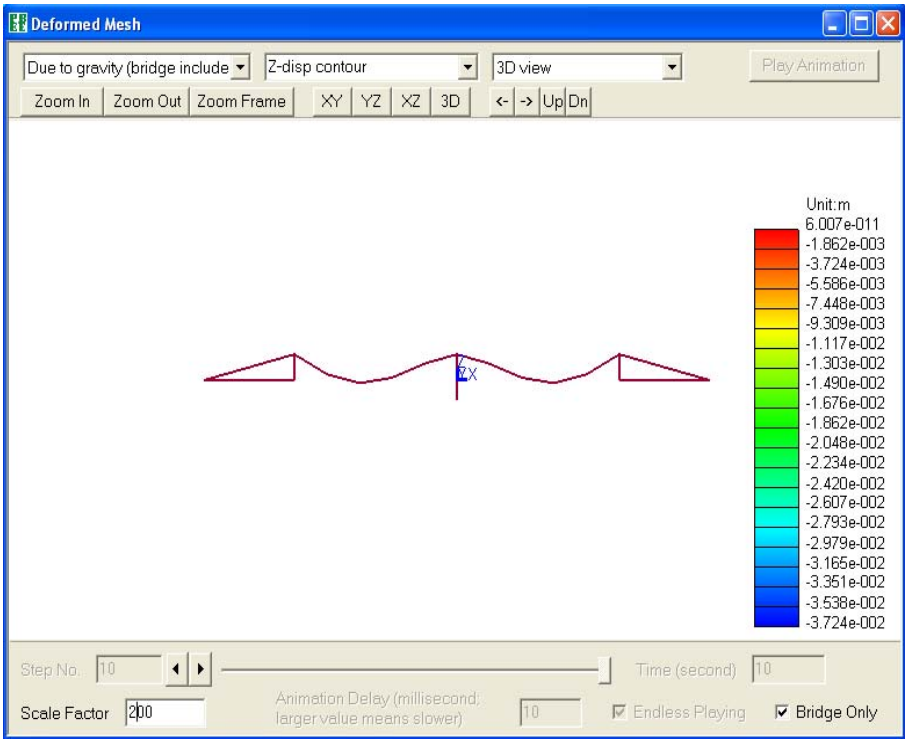


Fig. 122. Deck deformation under gravity (the maximum displacement is 0.0372 m)

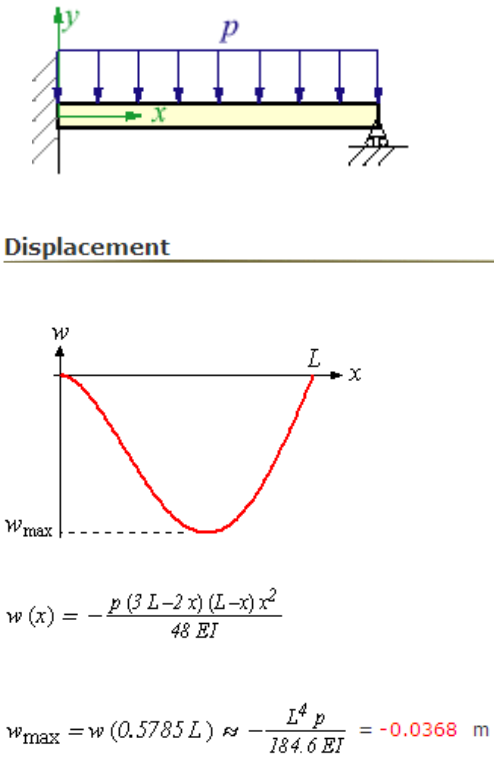


Fig. 123. Fixed end-roller beam analytical solution (from efunda.com)

4. Nonlinear-Column Bridge Pushover

a) Longitudinal Pushover (Fig. 124)

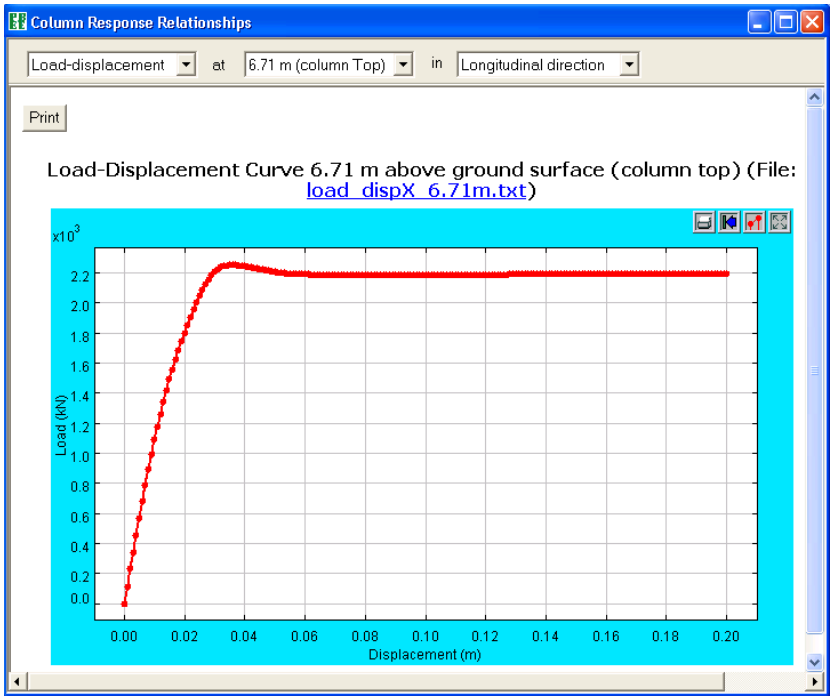


Fig. 124. Longitudinal pushover

b) Transverse Pushover (Fig. 125)

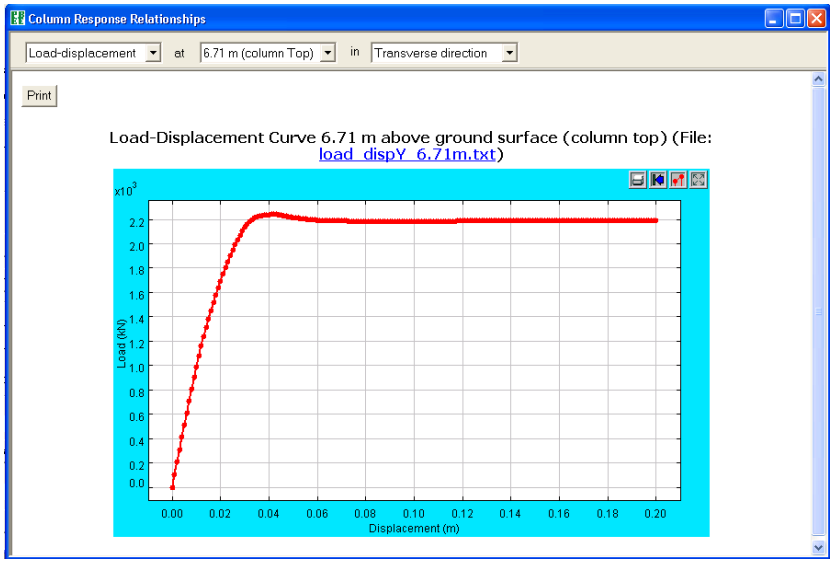


Fig. 125. Transverse pushover

10 Appendix C: How to Incorporate User-defined Motions

1) Directory Structure of a PBEE Motion Set

To conduct a PBEE analysis, input motions must be defined (please follow the steps shown in Fig. 50). The window to define PBEE input motions is shown in Fig. 51. Click **Browse** to select a PBEE motion set (Fig. 126). Click on the motion set name (e.g., PBEEMotionSet1) and then click on **OK** to choose this motion set (Fig. 126).

In BridgeBPEE, the input motions are organized in a format that the program can read. Specially, the input ground motions are sorted into bins. Fig. 127 shows the directory structure of a PBEE motion set named **PBEEMotionSet1**. The second level directories are bins (e.g., LMLR, LMSR, Near, SMLR, SMSR; see Fig. 126 and Fig. 127). The third level directories are earthquake names (e.g., there are 3 earthquakes under bin LMLR: BORREGO, LOMAP, NORTHR; see Fig. 127). And the fourth level directories are the input motion names (e.g., there is 1 input motion under earthquake BORREGO: A-ELC; see Fig. 127).

Each motion is composed of 3 perpendicular acceleration time history components (2 laterals and one vertical). As shown in Fig. 127, each motion folder contains 6 files categorized into 2 file types: the DATA files contain the time history (acceleration unit in g) of a component and the INFO files contain the characteristics of the corresponding component. Fig. 128 and Fig. 129 displays sample INFO & DATA files. Naming of these files has to follow the format below: Input motion name + angle (or "–UP" or "–DWN" for vertical component) + ".AT2" + ".data" (or ".info")

Note that the filenames with the smaller angle will be used for the longitudinal direction and the other one (with the larger angle) will be used for the transverse direction.

The first 2 lines of each INFO file must follow the style of the example below:
{Data points NPTS}{4000}
{Sampling period DT (sec)}{0.01}

Where 4000 and 0.01 are the number of data points, and the time step, respectively, of an input motion component.

2) Steps to Create an Input Motion

Based on the above description for the directory structure of a PBEE motion set, one can easily create an input motion (Fig. 130):

Step 1: create a folder and rename to your PBEE motion set name (e.g. MotionSet1; see Fig. 130).

Step 2: create a folder under the motion set folder and rename to your bin name (e.g., bin1).

Step 3: create a folder under the bin folder and rename to your earthquake name (e.g., Quake1).

Step 4: create a folder under the earthquake name and rename to your input motion name (e.g. MOTION1).

Step 5: create the 6 files (3 INFO files and 3 DATA files) for this input motion (Fig. 130).

Note: If you download the input motion files from the PEER NGA Database, there is no need to re-format the data into one column as shown in Fig. 129. Just copy the data points into the corresponding DATA files. And then make the INFO files containing the number of data points and the sampling period DT (2 lines) according to the header information.

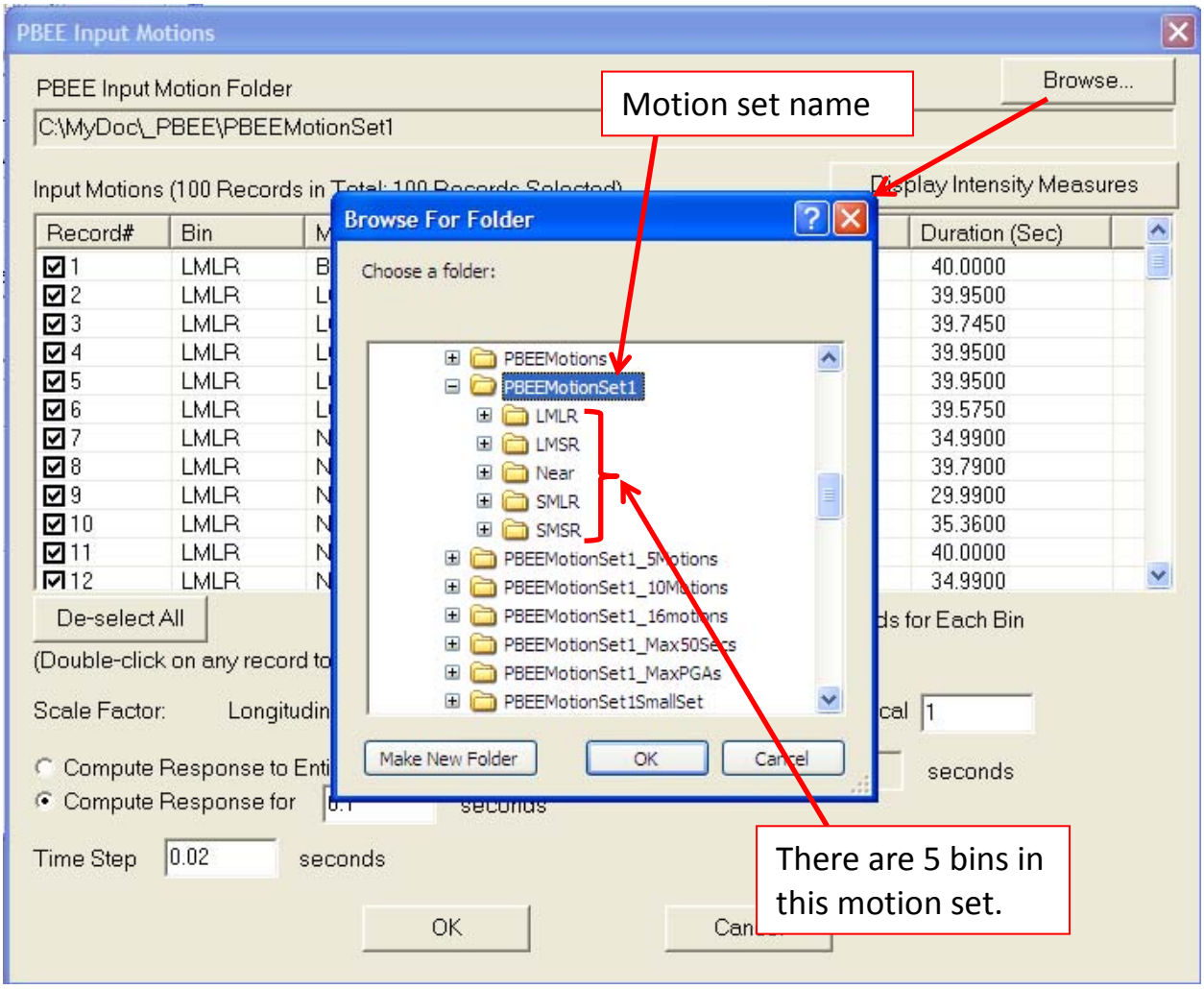


Fig. 126. Choosing PBEE motion set

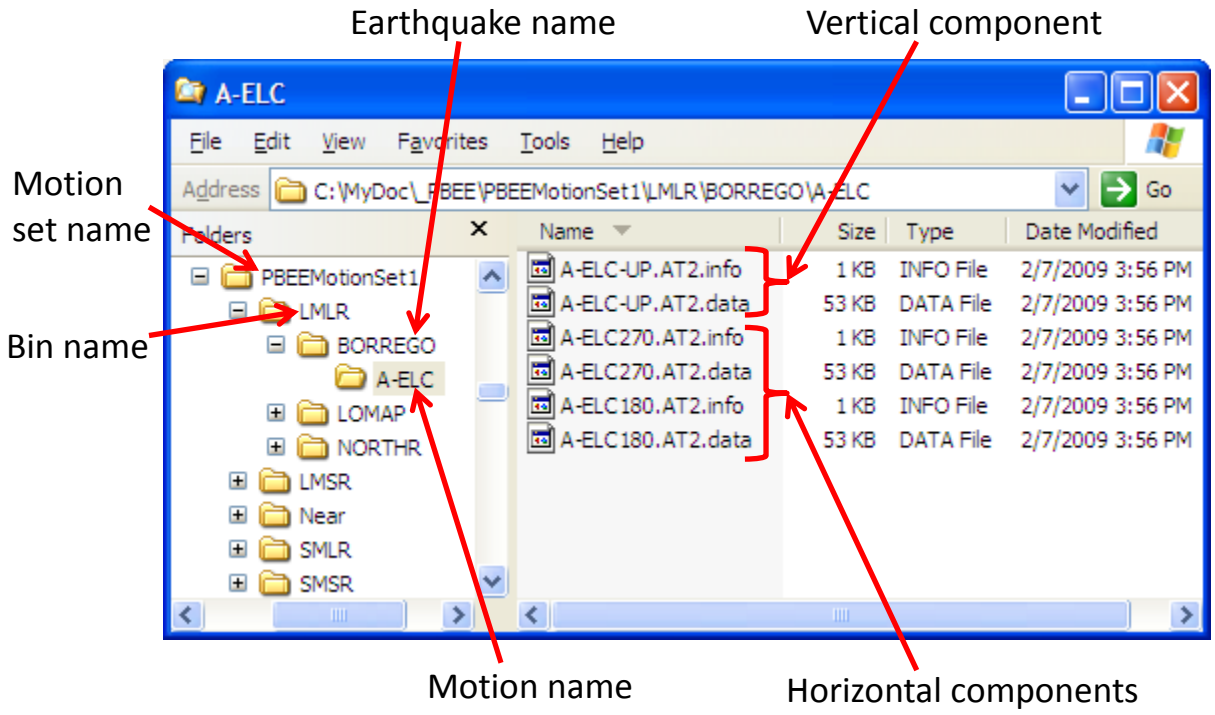


Fig. 127. Directory structure of PBEE motion set

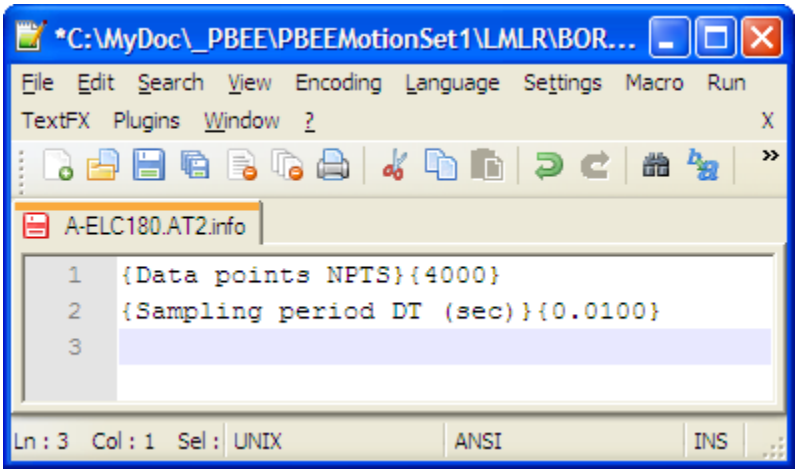

*C:\MyDoc_PBEE\PBEEMotionSet1\LMLR\BOR...
File Edit Search View Encoding Language Settings Macro Run
TextFX Plugins Window ?
A-ELC180.AT2.info
1 {Data points NPTS}{4000}
2 {Sampling period DT (sec)}{0.0100}
3
Ln : 3 Col : 1 Sel : UNIX ANSI INS

Fig. 128. Sample .info file

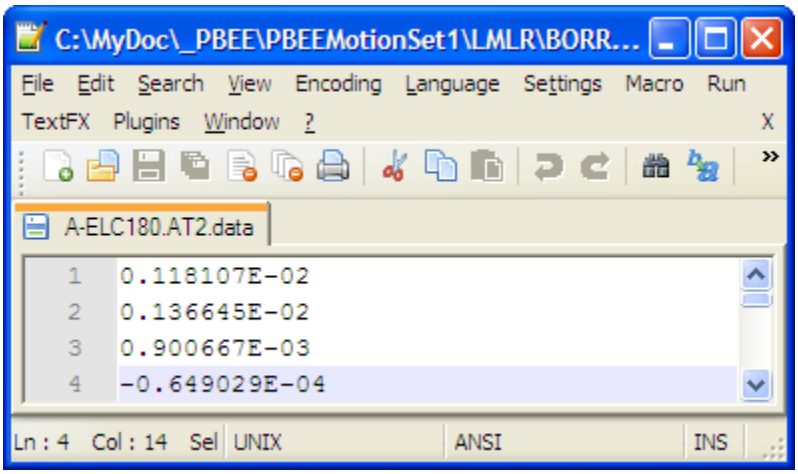


Fig. 129. Sample .data file

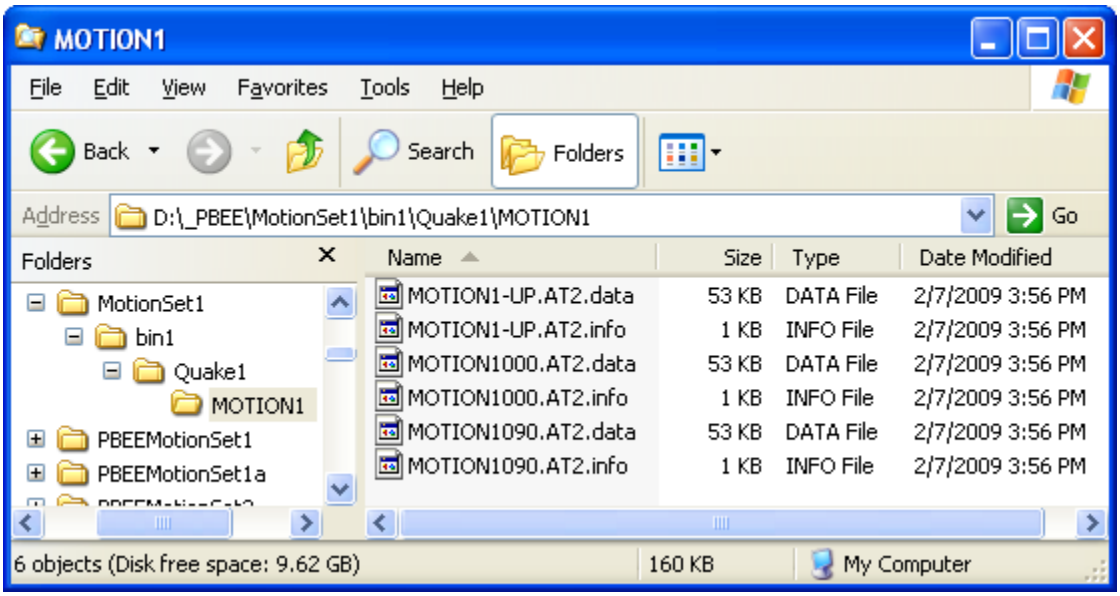

MOTION1
File Edit View Favorites Tools Help
Back Search Folders
Address D:_PBEE\MotionSet1\bin1\Quake1\MOTION1
Go
Folders
MotionSet1
bin1
Quake1
MOTION1
PBEEMotionSet1
PBEEMotionSet1a
Name Size Type Date Modified
MOTION1-UP.AT2.data 53 KB DATA File 2/7/2009 3:56 PM
MOTION1-UP.AT2.info 1 KB INFO File 2/7/2009 3:56 PM
MOTION1000.AT2.data 53 KB DATA File 2/7/2009 3:56 PM
MOTION1000.AT2.info 1 KB INFO File 2/7/2009 3:56 PM
MOTION1090.AT2.data 53 KB DATA File 2/7/2009 3:56 PM
MOTION1090.AT2.info 1 KB INFO File 2/7/2009 3:56 PM
6 objects (Disk free space: 9.62 GB) 160 KB My Computer

Fig. 130. Example of user-defined motion

11 Appendix D: Calculation of Steel and Concrete Material Properties

Steel Bars

By default, the Steel02 material is used to simulate steel bars. The format of the Steel02 command is as follows (Mazzoni et al. 2009):

uniaxialMaterial Steel02 $matTag $fy $E0 $b $R0 $cR1 $cR2

Where $fy is the steel yield strength (Table 2), $E0 is Young's modulus of steel, and $b is the strain-hardening ratio (ratio between post-yield tangent and initial elastic tangent), $R0, $cR1 and $cR2 are parameters to control the transition from elastic to plastic branches.

The number of longitudinal bars is calculated as follows:

$$\#bars = \frac{\rho_s A_c}{A_b} \tag{5}$$

Where ρ_s is the longitudinal steel percentage (Table 1), A_c the column cross-section area, A_b is the cross-section area of the steel bar.

If the number of longitudinal bars is known, the longitudinal steel percentage (reinforcement ratio) can be calculated:

$$\rho_s = \frac{A_s}{A_c} \tag{6}$$

Where A_s is the area of longitudinal steel, which is equal to the area of each bar times the number of bars. For example, the diameter of a #18 bar is 2.257 inches, so area is 4 in^2. If there are 10 bars in a 36 inch diameter circular column, then

$$\rho_s = \frac{10(4)}{\frac{\pi}{4}36^2} = 0.039$$

or 3.9%.

The transverse steel percentage (reinforcement ratio) for a spirally confined circular column, currently the only type of column supported in the interface, is

$$\rho_t = \frac{\pi(d_{db}{}^2)}{s(d_{cc})} \tag{7}$$

Where d_{bt} is the diameter of the transverse spiral (always smaller than the diameter of the longitudinal bars). The spacing between transverse bars is *s*. The diameter of the confined core is d_{cc} which is the gross diameter minus twice the cover and minus the diameter of the transverse bars (see Eq. 10). So for a #5 spiral spaced at 3 inches on center in the same column mentioned above.

$$\rho_t = \frac{\pi(\frac{5}{8})^2}{3(36-2(2)-\frac{5}{8})} = 0.013 \tag{8}$$

or 1.3%.

Currently the transverse reinforcement does affect the shear response (through changes in the uniaxial constitutive model for the concrete core). However, the columns are modeled considering only flexurally dominated response (i.e., there is no accounting for shear flexibility or shear degradation directly). Additional relevant details on the parameters used in both the Cover and Core Concrete are included in Appendix D (below).

Cover concrete

The Concrete02 material is used to simulate the concrete (for both cover and core). The format of the Concrete02 command is as follows:

uniaxialMaterial Concrete02 $matTag $fpc $epsc0 $fpcu $epsu $lambda $ft $Ets

Where $fpc is the concrete compressive strength, $epsc0 is the concrete strain at maximum strength, $fpcu is the concrete crushing strength, $epsu is the concrete strain at crushing strength (all of the above values are entered as negative), $lambda is the ratio between unloading slope at $epsu and initial slope, $ft is the tensile strength, and $Ets is tension softening stiffness (absolute value) (slope of the linear tension softening branch).

For cover concrete, $fpc is equal to the concrete unconfined strength in Table 1, $epsc0 = 0.002, $fpcu = 0.0, $epscu = 0.006, $lambda = 0.1, $ft = (0.14)$fpc, and $Ets = $ft / $epsc0.

Core concrete

i) For core concrete of circular column cross sections according to the Mander model, the procedure to calculate the confined concrete strength $fpc ($= f_{cc}$) is as follows:

$$f_{cc} = f_c^{'}(-1.254 + 2.254\sqrt{1 + 7.94\frac{f_e^{'}}{f_c^{'}}} - 2\frac{f_e^{'}}{f_c^{'}}) \quad (9)$$

Where $f_c^{'}$ is the unconfined compressive strength and $f_e^{'}$ can be obtained from the following equation:

$$f_e^{'} = \frac{1}{2} K_e \rho_t f_y \quad (10)$$

Where f_y is the steel yield strength, ρ_t is the transverse steel percentage, and K_e can be obtained from the following equation for spirally confined circular columns:

$$K_e = \frac{(1 - \frac{S'}{2d_{cc}})^2}{(1 - \rho_{cc})} \quad (11)$$

Where:

$$\rho_{cc} = \frac{A_s}{A_{cc}} \quad (12)$$

An assumed value of the area of the confined core is used for default values. This area should be modified based on the expected compressive block in the column during lateral loading.

$$A_{cc} = \frac{\pi (d_{cc})^2}{4} \quad (13)$$

$$S^{'} = \frac{\pi d_{bt}^{\ 2}}{\rho_t d_{cc}} \quad (14)$$

Where d_{bt} is the transverse bar diameter

$$d_{cc} = D_L - 2c - d_{bt} \quad (15)$$

Where c is the clear cover (c = 1.5")

ii) $epsc0

$$epsc0 = \frac{2 f_{cc}}{E_c} \quad (16)$$

Where:

$$Ec = 0.043w^{1.5}\sqrt{f_c^{'}} \tag{17}$$

Where w is the concrete unit weight (unit: kg/m^3)

iii) \$epsu (= *epscu*)

$$epscu = 0.004 + \varepsilon_s \frac{f_y}{f_c^{'}} \rho_t \tag{18}$$

Where ε_s is the ultimate steel strain ($\varepsilon_s = 0.12$)

iv) \$fpcu ($= f_{cu}$)

$$f_{cu} = \frac{f_{cc}(epscu)}{(epsc)}\left(\frac{(epscr)}{(epscr) - 1 + \left(\frac{(epscu)}{(epsc)}\right)^{(epscr)}}\right) \tag{19}$$

Where:

$$epsc = (epsc0)\left(1 + 5\left(\frac{f_{cc}}{f_c^{'}} - 1\right)\right) \tag{20}$$

$$epscr = \frac{E_c}{E_c - \frac{f_{cc}}{(epsc)}} \tag{21}$$

Notes:

1. The information above is specific to the Steel02 and Concrete02 models of the Fiber section. Other options include (Fig. 10), Steel01 and Concrete01 (for more information please see the OpenSees documentation), and Elastic properties for the fibers. These options can be activated by clicking on the default Steel02 or Concrete02 sections (Fig. 10) and changing these options.

2. A different property may be specified for the Column below grade (for instance to roughly represent a large pile group as a large single column). If this option is selected (Fig. 7), the column below grade will have linear properties as specified by its diameter and Young's Modulus).

3. All the equations presented in this Appendix are based on the Mander model for spiral-reinforced circular concrete columns. The user may want to use their own constitutive model or parameters. In this case, the values of these parameter can be defined directly in Fig. 10.

12 Appendix E: Customization of PBEE Quantities

Users can customize PBEE quantities through updating a file named PBEE.DLL which is located at the installation folder (C:\Program Files\BridgePBEE or C:\Program Files(x86)\BridgePBEE on a 64bit PC). Please follow the steps below to build an updated PBEE.DLL file and then replace the one at the installation folder.

Step 1: Download PBEE.ZIP

Please go to the BridgePBEE website to download a source code project file (filename: PBEE.zip) for Visual Studio. We'll use this one to build the PBEE.DLL file.

Step 2: Open PBEE.SLN File

Unzip PBEE.zip to a certain location and then use Visual Studio (2005 version preferred) to open a Visual Studio Solution file named PBEE.SLN (Fig. 131).

Open the file named PBEE.CPP and make appropriate changes (Fig. 132).

Step 3: Build PBEE.DLL File

Under Visual Studio 2005, click menu Build and then Build Solution to build an updated PBEE.DLL file (Fig. 133).

Step 4: Replace PBEE.DLL File

Make sure that BridgeBEEE is not running and then copy the new PBEE.DLL file to the installation folder and overwrite the old one (Fig. 134).

Step 5: Run BridgePBEE

Start BridgePBEE, the program is now running with the updated PBEE quantities.

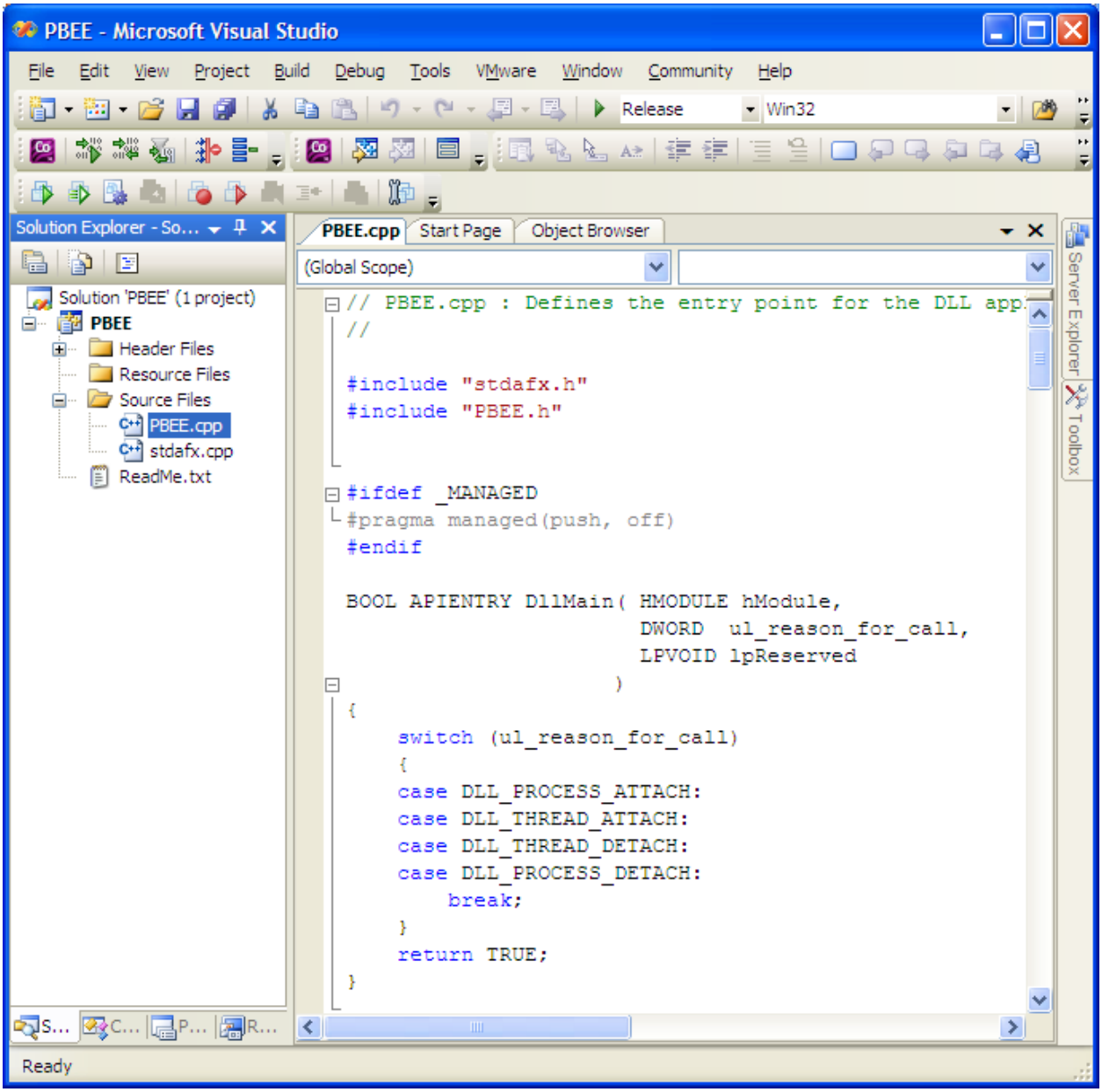


Fig. 131. Visual Studio file PBEE.SLN

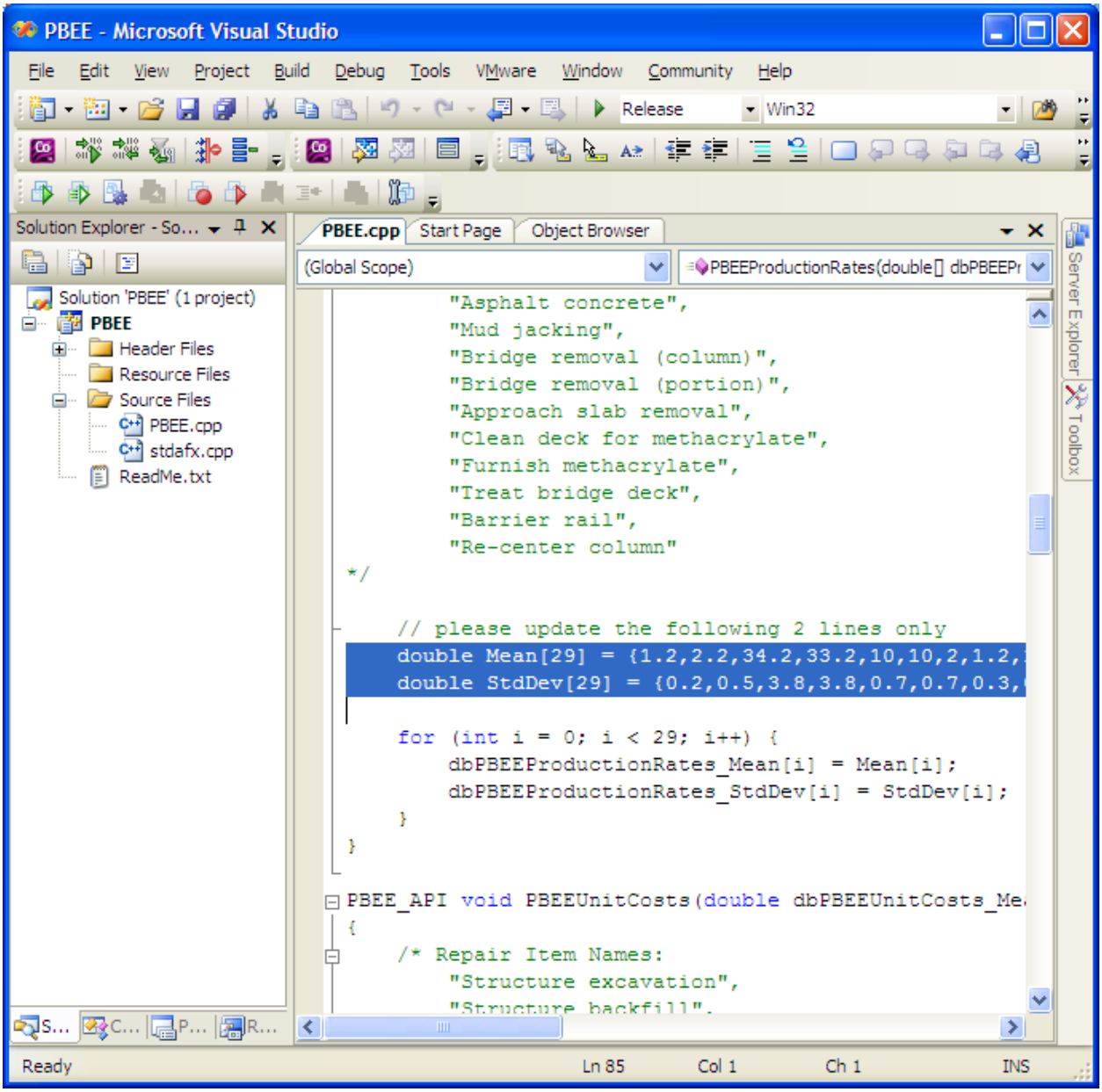


Fig. 132. Modifying file PBEE.CPP

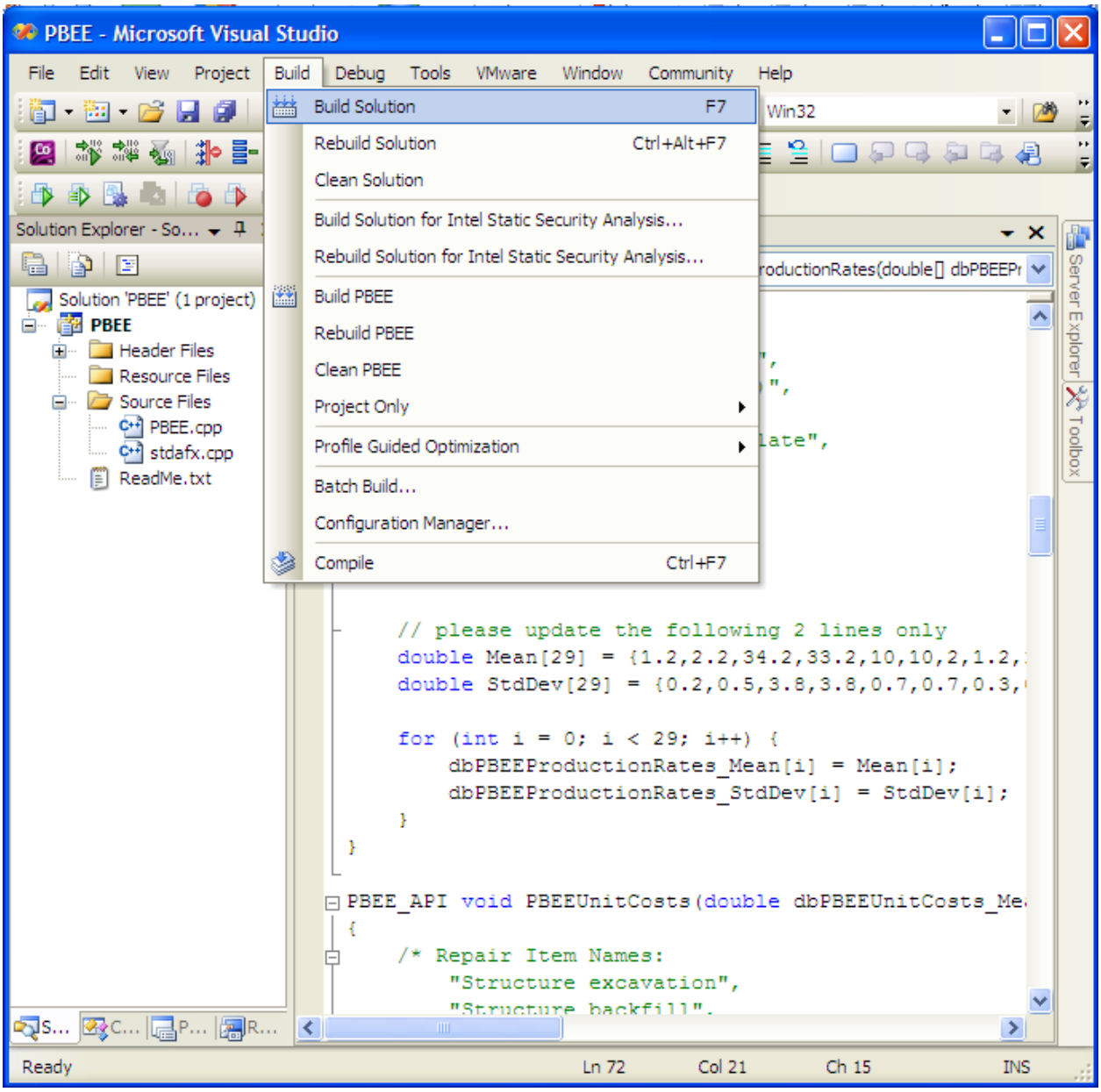


Fig. 133. Building PBEE.DLL in Visual Studio 2005

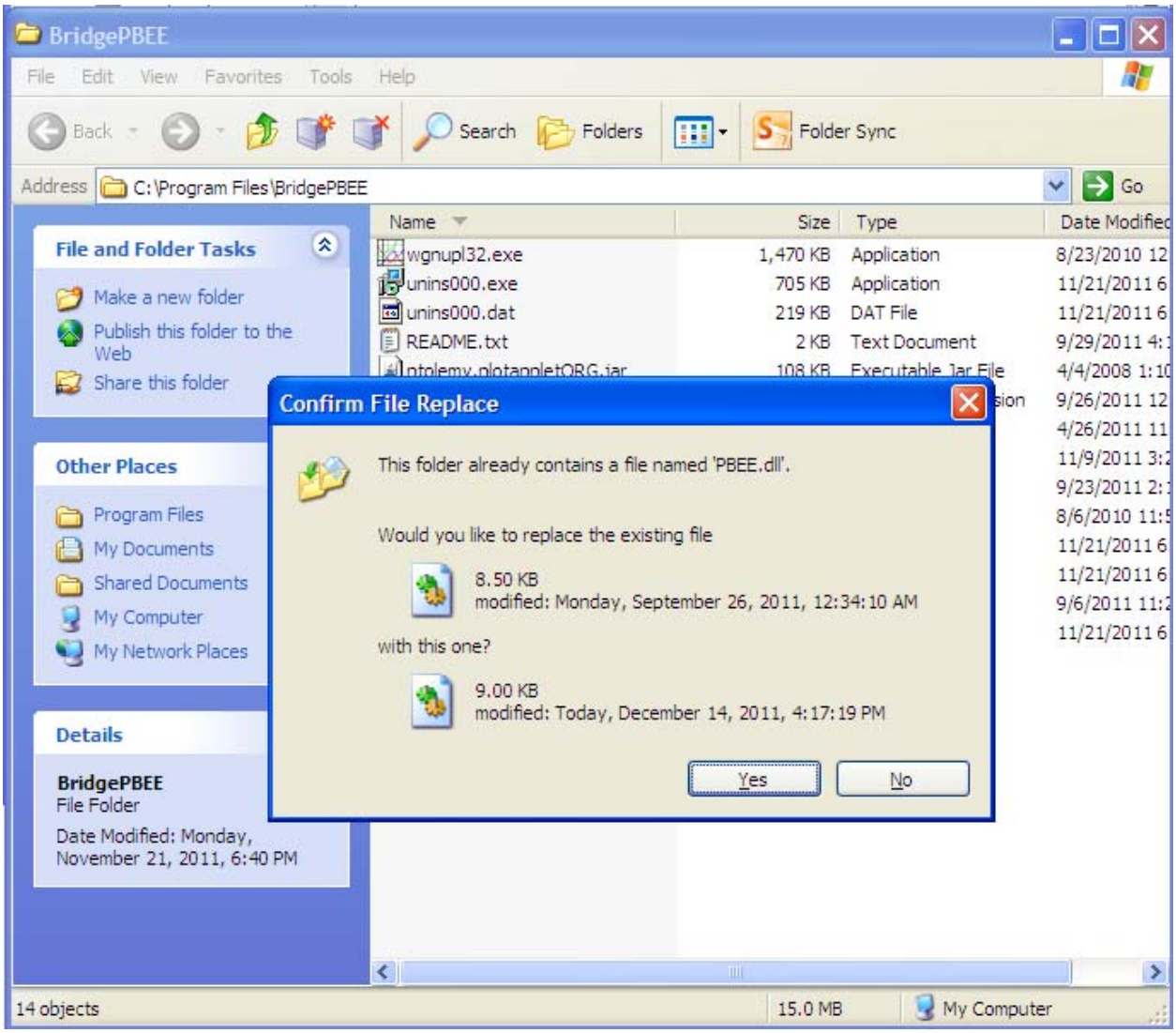


Fig. 134. Replacing file PBEE.DLL under the installation folder

13 References

Aviram, A., Mackie, K.R., and Stojadinovic, B. (2008). Effect of Abutment Modeling on the Seismic Response of Bridge Structures, *Earthquake Engineering and Engineering Vibration* 7(4), 395-402.

Berry, M.P., and Eberhard, M.O. (2007). Performance Modeling Strategies for Modern Reinforced Concrete Bridge Columns, *Report No. 2007/07*, Pacific Earthquake Engineering Research Center, University of California, Berkeley, California.

Kotsoglu, A., and Pantazopoulou, S. (2006). Modeling of Embankment Flexibility and Soil-structure Interaction in Integral Bridges, *Proceedings of First European Conference on Earthquake Engineering and Seismology*, September 3–8, Geneva, Switzerland.

Caltrans SDC, 2004. Caltrans Seismic Design Criteria, Version 1.3, California Department of Transportation, Sacramento, California.

Caltrans SDC, 2010. Caltrans Seismic Design Criteria, Version 1.6, California Department of Transportation, Sacramento, California.

Elgamal, A., Yang, Z., Parra, E., and Ragheb, A. 2003. Modeling of cyclic mobility in saturated cohesionless soils. International Journal of Plasticity, 19(6), 883-905.

Elgamal, A., and Lu, J. 2009. A Framework for 3D finite element analysis of lateral pile system response, Proceedings of the 2009 International Foundation Congress and Equipment Expo, Contemporary Topics in In Situ Testing, Analysis, and Reliability of Foundations, ASCE GSP 186, M. Iskander, D. F. Laefer, and M. H. Hussein, Editors, Orlando, Florida, March 15–19, pp. 616-623.

Elgamal, A., Jinchi Lu, J., and Forcellini, D. 2009a. Mitigation of liquefaction-induced lateral deformation in a sloping stratum: 3D numerical simulation, Journal of geotechnical and geoenvironmental engineering, ASCE, Vol. 135, No. 11, November , 1672–1682.

Elgamal, A., Lu, J., Yang, Z. and Shantz, T. 2009b. Scenario-focused three-dimensional computational modeling in geomechanics, Alexandria, Egypt, October 3-5, 4 iYGEC'09 - 4th International Young Geotechnical Engineers' Conference, 2 - 6 October, ISSMGE.

Elgamal, A. (2010). Calibrated 3D computational modeling of soil-structure systems and liquefaction scenarios, Proc. Fifth Intl. Conf. on Recent Advances in Geotechnical Earthquake Engineering and Soil Dynamics, May 24-29, San Diego, CA.

Lu J., 2006. Parallel finite element modeling of earthquake site response and liquefaction, PhD Thesis, Department of Structural Engineering, University of California, San Diego, La Jolla, CA.

Mackie, K.R., and Stojadinovic, B. (2005). "Fragility basis for California highway overpass bridge seismic decision making." *PEER Report No. 2005/02*, Pacific Earthquake Engineering Research Center, University of California, Berkeley.

Mackie, K. and Stojadinović, B. (2006). Seismic Vulnerability of Typical Multi-span California Highway Bridges, *Proceedings of the Fifth National Seismic Conference on Bridges and highways*, September 18–20, San Francisco.

Mackie, K.R., Wong, J-M., and Stojadinovic, B. (2008). "Integrated probabilistic performance-based evaluation of benchmark reinforced concrete bridges." *PEER Report No. 2007/09*, Pacific Earthquake Engineering Research Center, University of California, Berkeley.

Mackie, K.R., Wong, J-M., and Stojadinovic, B. (2010). Post-earthquake bridge repair cost and repair time estimation methodology. *Earthquake Engineering and Structural Dynamics*, 39(3): 281-301.

Mackie, K.R., Wong, J-M., and Stojadinovic, B. (2011). Bridge damage and loss scenarios calibrated by schematic design and cost estimation of repairs. *Earthquake Spectra*, 27: 1127-1145.

Maroney, B.H., and Chai, Y.H. (1994). Seismic Design and Retrofitting of Reinforced Concrete Bridges, *Proceedings of 2nd International Workshop, Earthquake Commission of New Zealand*, Queenstown, New Zealand.

Mander, J.B., Priestley, M.J.N., and Park, R. (1988). Theoretical Stress–Strain Model for Confined Concrete, *Journal of the Structural Division ASCE,* **114**, pp. 1804–1826.

Mazzoni, S., McKenna, F., Scott, M. H., Fenves, G. L., et al. (2009). Open System for Earthquake Engineering Simulation, User Command-Language Manual, Pacific Earthquake Engineering Research Center, University of California, Berkeley, OpenSees version 2.0, May.

Megally, S.H., Seible, F., Bozorgzadeh, A., Restrepo, J. and Silva, P.F. (2003). Response of Sacrificial Shear Keys in Bridge Abutments to Seismic Loading, *Proceedings of the FIB Symposium on Concrete Structures in Seismic Regions*, May 6-9, Athens, Greece.

Shamsabadi, A., Rollins, K.M, and Kapuskar, M. (2007). Nonlinear Soil–Abutment–Bridge Structure Interaction for Seismic Performance-Based Design, *Journal of Geotechnical and Geoenvironmental Engineering*, 133(6), 707–720, June.

Shamsabadi, A., Khalili-Tehrani, P., Stewart, J.P., and Taciroglu, E. (2010). Validated Simulation Models for Lateral Response of Bridge Abutments with Typical Backfills, *J. Bridge Eng.*, 15(3), 302-311, May.

Werner, S.D. (1994). Study of Caltrans' Seismic Evaluation Procedures for Short Bridges, *Proceedings of the 3rd Annual Seismic Research Workshop*, Sacramento, California.

Yang, Z., Elgamal, A., and Parra, E. 2003. A computational model for cyclic mobility and associated shear deformation, *Journal of Geotechnical and Geoenvironmental Engineering*, 129 (12), 1119-1127.

Zhang, J., and Makris, N. (2002). Kinematic Response Functions and Dynamic Stiffnesses of Bridge Embankments, *Earthquake Engineering & Structural Dynamics*, **31**(11), pp. 1933–1966.